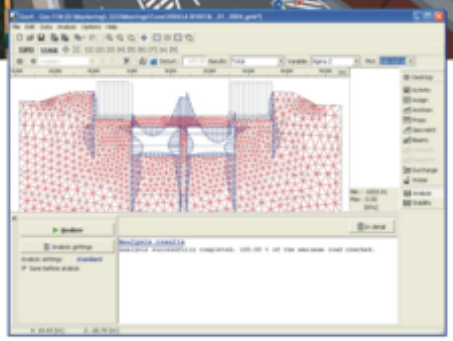
geotechnical software suite

GEO5

User´s Guide

version 5.3



fine civil engineering software

www.fine.cz

Table of Contents

Using help 7
Using function Find 8
User defined environment 8
Window for application 9
Control menu 9
Horizontal tool bars 10
Tool bar Files 10
Tool bar Scale and shift 11
Tool bar Plot setting 12
Tool bar Stage of construction 12
Vertical tool bars 13
Setting visualization style 13
Style manager 15
Frames 15
Tables 17
Dialogue windows 18
Active dimensions and objects 19
Unit metric / imperial 20
Copy to clipboard 20
Options 20
Options – copy to clipboard 21
Options – print picture 22
Options - input 23
Common input 23
Project – Earth pressures 23
Inputting and editing soils 24
Soil classification 26
Soil and rock labels 27
Manual classification of soil 27
Interfaces in 2D environment 28
Adding interface 29
Editing interface 30
Corrector of inputted interface 30
World coordinates 32
Assigning soils 32
Design coefficients 33
Running more analyses / verifications 34
Connecting programs 35
Selecting and storing views 36
Setting results visualization 37
Setting color range 38
Scale color definition 39
Input regimes and analysis 40
Program Earth Pressure 40
Project 40
Geometry 40
Profile 41
Soils 42
Assign 43
Terrain 43
Water 44
Surcharge 45
Earthquake 46
Setting 47
Analysis 47
Program Sheeting Design 48
Project 48
Profile 49
Soils 49
Assign 50
Geometry 51
Anchors 51

Props 52
Supports 52
Pressure specification 53
Terrain 54
Water 55
Surcharge 56
Forces 57
Earthquake 57
Setting 58
Analysis 59
Program Sheeting Check 60
Project 60
Profile 61
Modulus of subsoil reaction 61
Soils 62
Geometry 63
Adding and editing section 63
User catalogue 64
Assign 65
Excavation 66
Terrain 67
Water 68
Surcharge 69
Forces 70
Anchors 70
Props 71
Supports 72
Earthquake 73
Setting 74
Analysis 74
Internal stability 76
External stability 77
Envelopes 78
Program Slope Stability 78
Project 78
Interface 79
Embankment 79
Eart cut 80
Soils 81
Rigid body 82
Assign 82
Anchors 83
Reinforcements 84
Surcharge 85
Water 86
Earthquake 87
Setting 88
Analysis 88
Constrains on the optimization procedure 90
Height multiplier 91
Program Cantilever Wall 92
Project 92
Geometry 93
Material 93
Profile 94
Soils 95
Assign 96
Terrain 96
Water 97
Surcharge 98
Front face resistance 99
Inputted forces 100
Earthquake 101
Base anchorage 101
Setting 102

Verification 103
Bearing capacity 104
Dimensioning 104
Stability 105
Program Masonry wall 106
Project 106
Geometry 107
Material 107
Profile 108
Soils 109
Assign 110
Terrain 110
Water 111
Surcharge 112
Front face resistance 113
Inputted forces 114
Earthquake 115
Base anchorage 115
Setting 116
Verification 117
Bearing capacity 118
Dimensioning 118
Stability 119
Program Gravity Wall 120
Project 120
Geometry 121
Material 121
Profile 122
Soils 123
Assign 124
Terrain 124
Water 125
Surcharge 126
Front face resistance 127
Inputted forces 128
Earthquake 129
Setting 129
Verification 130
Bearing capacity 130
Dimensioning 131
Stability 132
Program Block Wall 133
Project 133
Geometry 133
Profile 134
Soils 135
Assign 136
Terrain 136
Water 137
Surcharge 138
Front face resistance 139
Inputted forces 140
Earthquake 141
Setting 141
Verification 142
Bearing capacity 142
Dimensioning 143
Stability 144
Program Gabion 145
Project 145
Material 145
Geometry 146
Profile 147
Soils 148
Assign 148

Terrain 149
Water 150
Surcharge 150
Front face resistance 151
Inputted forces 152
Earthquake 153
Setting 153
Verification 154
Bearing capacity 154
Dimensioning 155
Stability 156
Program Spreadfooting 157
Project 157
Project - Analyses 157
Profile 158
Soils 159
Assign 160
Foundation 160
Load 161
Import of loading 162
Geometry 163
Sand-gravel cushion 164
Material 165
Surcharge 166
Water, incompressible subsoil 167
Setting 168
1.LS - bearing of a footing 169
2.LS - settlement and rotation of a footing 170
Dimensioning 171
Program Pile 172
Project 172
Profile 173
Modulus of subsoil reaction 173
Soils 174
Assign 175
Load 176
Geometry 176
Material 177
Water, incompressible subsoil 178
Negative skin friction 179
Setting 180
Vertical bearing capacity 180
Vertical bearing capacity CSN 180
Vertical bearing capacity FEM 181
Horizontal bearing capacity 182
Program Settlement 183
Project 183
Interface 183
Embankment 184
Eart cut 185
Incompressible subsoil 186
Soils 187
Assign 187
Surcharge 188
Water 189
Setting 190
Analysis 190
Program Svahovky 191
Projekt 191
Geometrie 192
Výztuhy 192
Základ 192
Profil 192
Zeminy 193
Přiřazení 193

Terén 194
Voda 195
Přitížení 196
Odpor na líci 197
Zadané síly 197
Zemětřesení 198
Nastavení 198
Posouzení 199
Únosnost 200
Dimenzace 201
Program Abutment 201
Project 202
Geometry cut 202
Wings 203
Geometry plane view 204
Foundation steps 205
Material 205
Profile 206
Soils 207
Loading - LC 208
Assign 208
Terrain 209
Water 210
Surcharge 211
Front face resistance 212
Inputted forces 213
Earthquake 214
Setting 214
Verification 215
Bearing capacity 216
Dimensioning 216
Stability 218
Program Nailed slopes 218
Project 218
Geometry 219
Types of nails 219
Geometry of nails 220
Material 221
Profile 221
Soils 222
Assign 222
Terrain 223
Water 224
Surcharge 225
Earthquake 226
Setting 226
Internal stability 227
Verification 227
Bearing capacity 228
Dimensioning 229
External stability 230
Outputs 231
Adding picture 231
Picture list 232
Print and export document 233
Print and export picture 234
Control menu Print and export 235
Tool bar Print and export 236
Setting header and footer 237
Page properties 238
Page numbering 239
About company 239

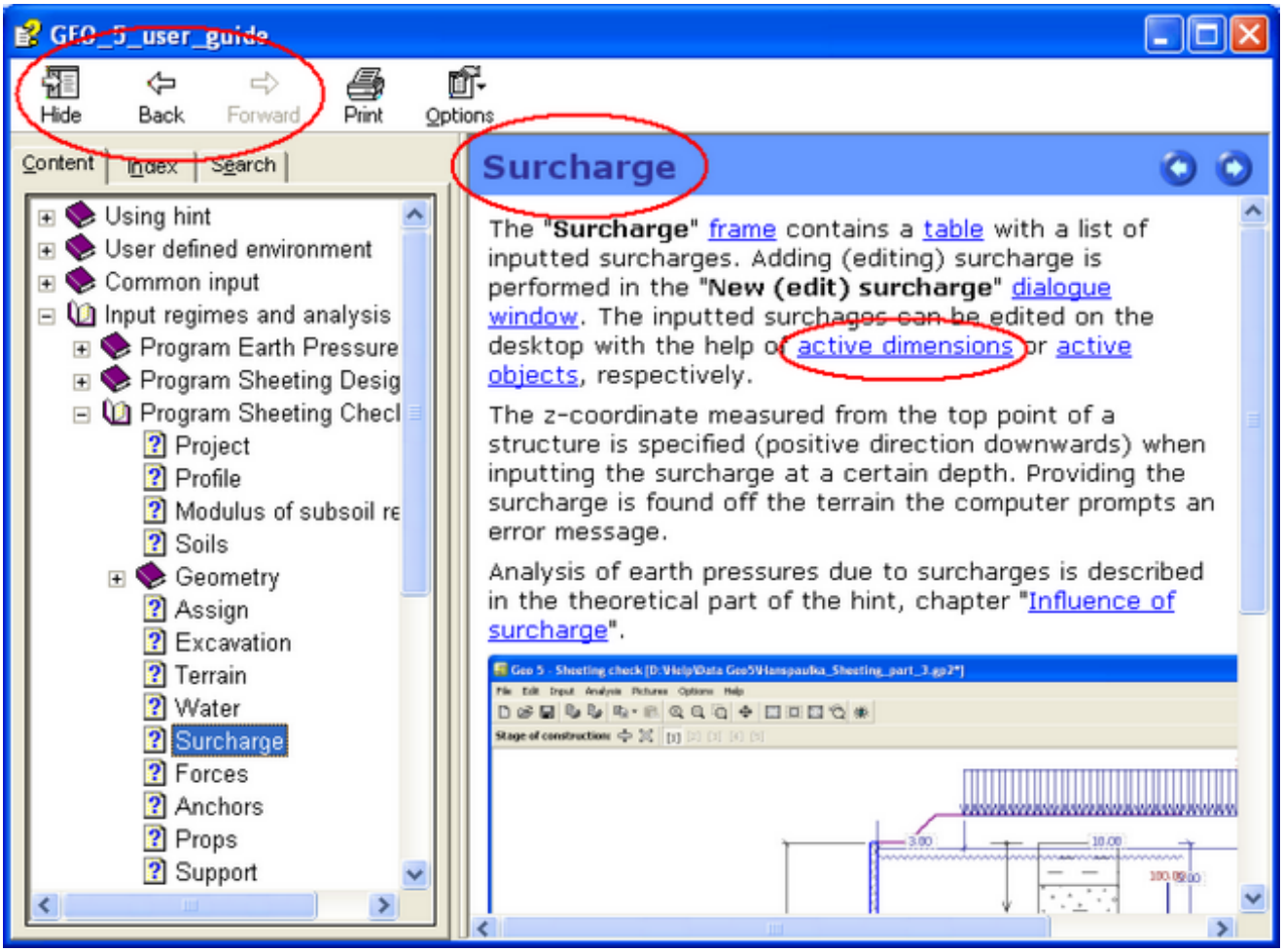
Using help

The text hint of all GEO programs is displayed in standard Windows EXPLORER viewer. The help dialogue window can be launched either through the program menu (items "Help", "Content") or using the Fn button "F1" anywhere in the program.
Some dialogue windows (e.g., "Add soil") allow for opening the corresponding chapter of the help by pressing the help button " ".

The dialogue window contains:

- Bar with basic tool buttons. The "Hide (Show)" button hides (shows) tree with a list help topics. The ("Back/Forward") buttons allow for listing pages, which have been already displayed. The "Print" button opens the printing dialogue window and the "Options" buttons opens the menu to set the EXPLORER window properties.
- Bar containing the tab sheet "Content" (shows tree with individual topics), tab sheets "Index" and "Search".
- "Tree" with a list hint topics – individual items in the tree are opened/closed by clicking the symbols "+"/"-" in front of the name.
- Window for displaying the help itself – the window header contains currently shown items and the "Back/Forward" buttons that function in the same way as already described buttons on tool bar.

The text of each help may further contain cross references to other items. The text of these references is underlined highlighted in blue color.

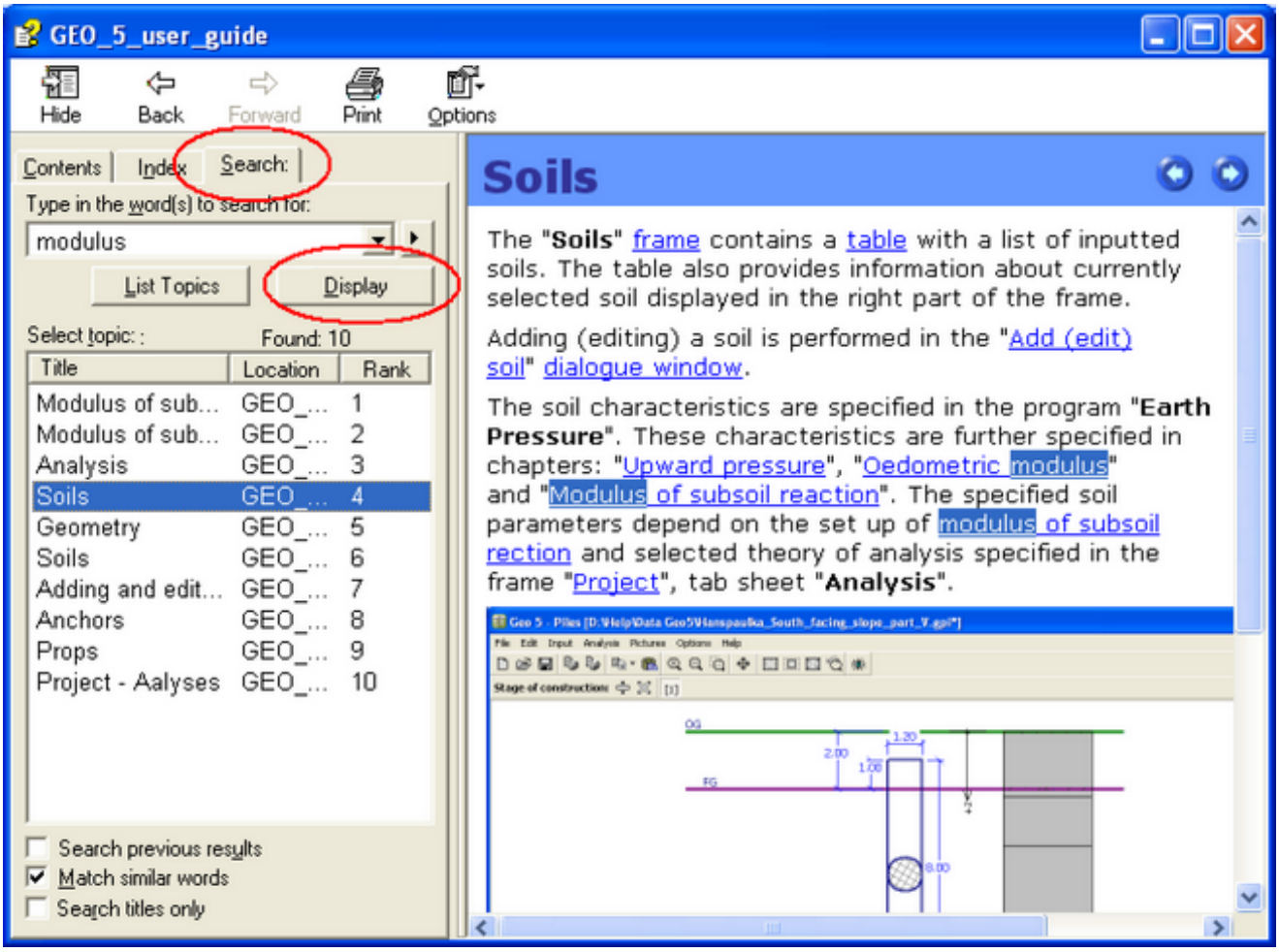

GEO_5_user_guide
Hide
Back
Forward
Print
Options
Content
Index
Search
Using hint
User defined environment
Common input
Input regimes and analysis
Program Earth Pressure
Program Sheeting Desig
Program Sheeting Checl
Project
Profile
Modulus of subsoil re
Soils
Geometry
Assign
Excavation
Terrain
Water
Surcharge
Forces
Anchors
Props
Support
Surcharge
The "Surcharge" frame contains a table with a list of inputted surcharges. Adding (editing) surcharge is performed in the "New (edit) surcharge" dialogue window. The inputted surcharges can be edited on the desktop with the help of active dimensions or active objects, respectively.
The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. Providing the surcharge is found off the terrain the computer prompts an error message.
Analysis of earth pressures due to surcharges is described in the theoretical part of the hint, chapter "Influence of surcharge".

Dialogue window "Help of GEO" - tab sheet "Contents"

Using function Search

Function "Search" allows for finding an arbitrary text in the subject help. The searched text is written into the field "Input searched text" and button "List of topics" lunches the search. The list of found topics containing the searched text is displayed in the colum n under the buttons. Clicking the mouse on the topic title and pressing the "Show" button displays the corresponding topic in the right part of the window (the double-clicking option is also available).

The searched text is highlighted in blue color. Switching back to "Content" tab sheet shows the topic location in the tree (help content).


GEO_5_user_guide
Hide
Back
Forward
Print
Options
Contents
Index
Search:
Type in the word(s) to search for:
modulus
List Topics
Display
Select topic:
Found: 10
Title
Location
Rank
Modulus of sub... GEO_... 1
Modulus of sub... GEO_... 2
Analysis GEO_... 3
Soils GEO_... 4
Geometry GEO_... 5
Soils GEO_... 6
Adding and edit... GEO_... 7
Anchors GEO_... 8
Props GEO_... 9
Project - Aalyses GEO_... 10
Search previous results
Match similar words
Search titles only
Soils
The "Soils" frame contains a table with a list of inputted soils. The table also provides information about currently selected soil displayed in the right part of the frame.
Adding (editing) a soil is performed in the "Add (edit) soil" dialogue window.
The soil characteristics are specified in the program "Earth Pressure". These characteristics are further specified in chapters: "Upward pressure", "Oedometric modulus" and "Modulus of subsoil reaction". The specified soil parameters depend on the set up of modulus of subsoil rection and selected theory of analysis specified in the frame "Project", tab sheet "Analysis".

Frame "Help" - tab sheet "Find"

User defined environment

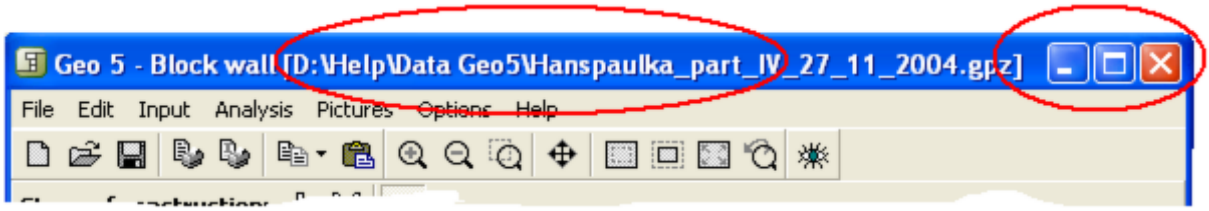
The programs GEO5 are standard windows applications. Managing the application environment (application window, dialogue windows, control menu, tables, frames, tool bars, copy to clipboard) applies to standard properties of the Windows environment. The programs support operating systems WIN 98, WIN 2000, WIN NT and WIN XP.

The minimum hardware and software configuration corresponding to class of PC PENTIUM III with 128 MB of operating memory and minimum resolution of graphic adapter and monitor of 1024x768 pixels with 256 bit color range is required. Recommended configuration corresponds to a class of PC computers of Pentium 4, 512 MB of operating memory.

Window for application

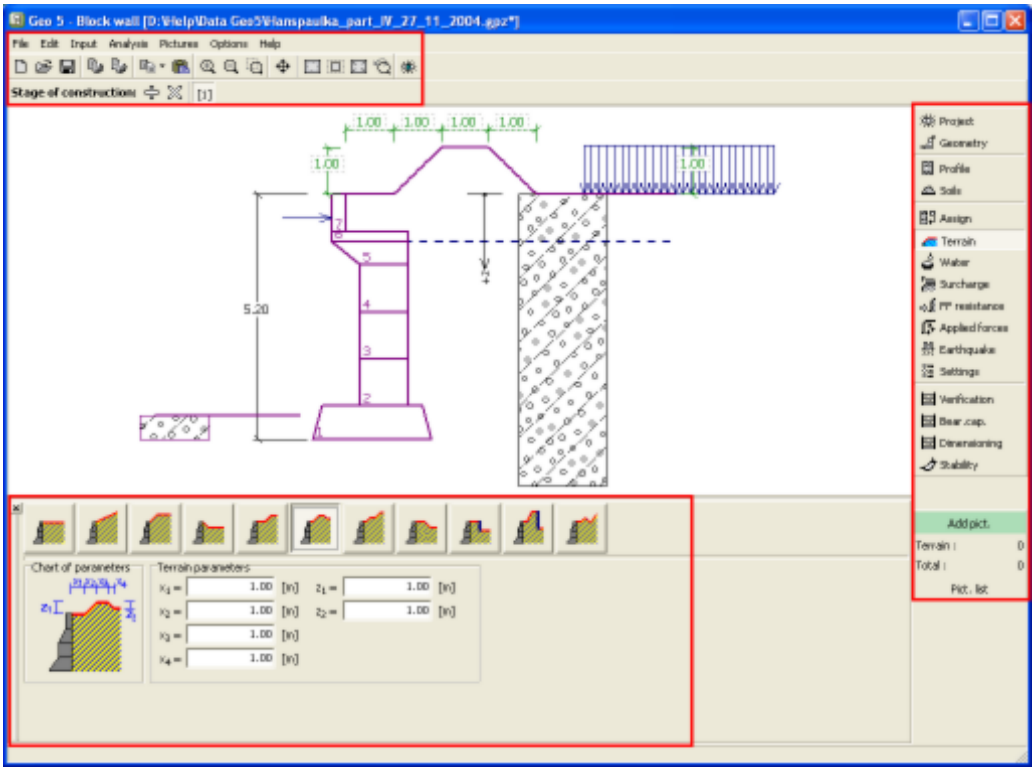
The program is launched in standard dialogue window containing all managing tools typical for the Windows environment (minimizing, maximizing and closing the application window...).

The window header displays information on currently executed task (file name and location) – see figure:



Managing tools of window for application

The desktop constitutes the window of application. It includes the control menu, horizontal tool bars, space for graphic visualization of the executed task and vertical tool bars to select individual inputting modes to specify the task. The bottom part of the desktop displays frames that allow the user to introduce various input parameters into the task. Location of individual elements on the desktop is evident from the following figure:



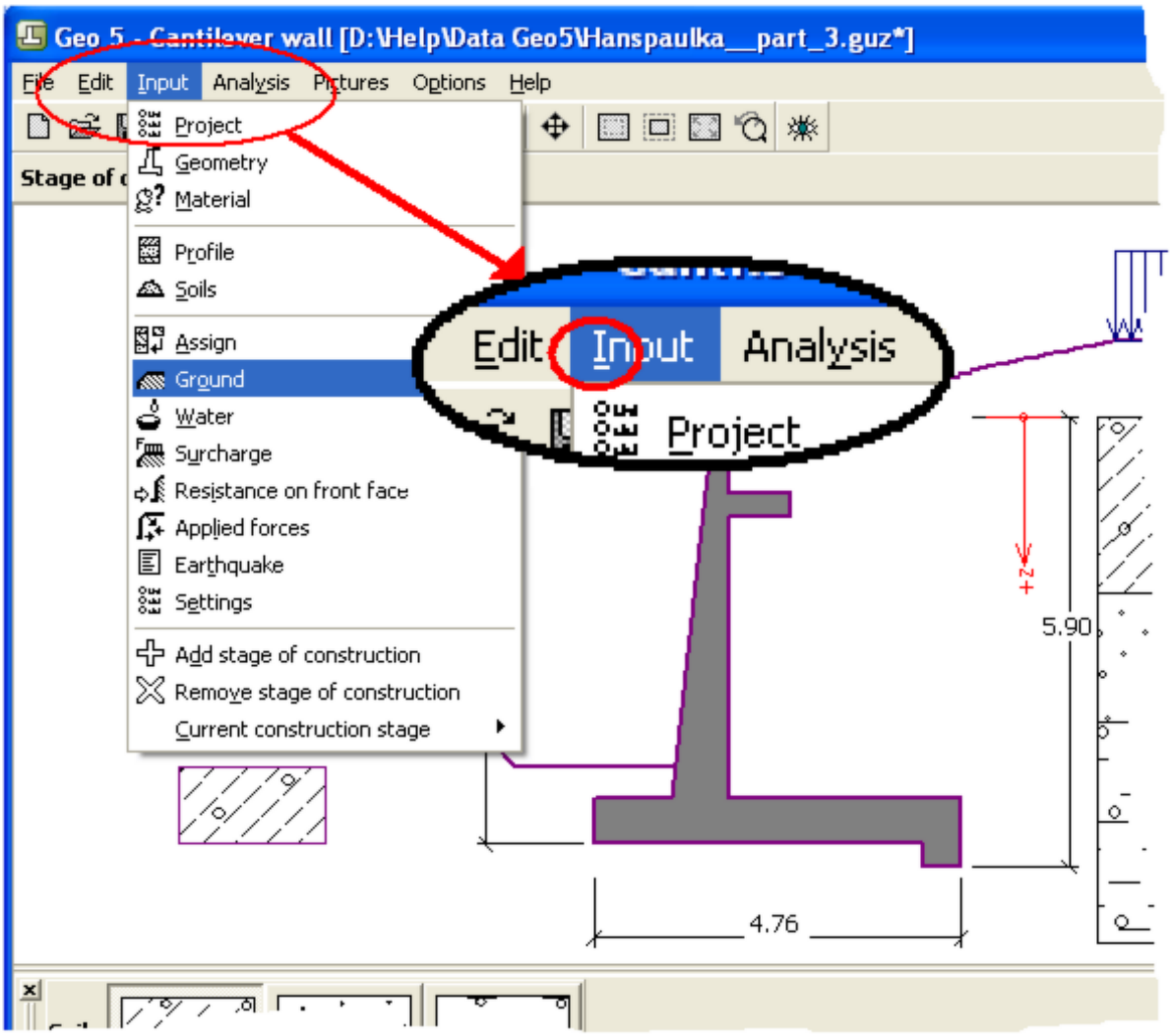
Managing tools of window for application

Control menu

Selecting an item from the menu is performed by clicking the left mouse button, or alternatively using keyboard by pressing ALT + underlined letter in the selected menu item.

As typical for the WINDOWS environment, some of the options in the menu can be replaced with the buttons on individual tool bars, or with abbreviated commands entered through the keyboard (providing it exists it is displayed next to the command in the menu – e.g., Save file – CTRL+S).

Some of the options in the program can be set only with the help of menu – e.g., program "Options".



Control menu of program

Horizontal tool bars

The program contains the following tool bars:

- Tool bar "Files"
- Tool bar "Scale and shift"
- Tool bar "Plot setting"
- Tool bar "Stage of construction"

Tool bar "Files"

The tool bar contains the following buttons:

Tool bar "Files"

Individual buttons function as follows:

New file

- opens a new file – if there is an existing task opened in the same window, the program prompts the user to save the unsaved data

Open file	• opens an existing file - if there is an existing task opened in the same window, the program prompts the user to save the unsaved data
Save data into file	• saves data of currently opened task – if no name is assigned to the task, the program opens the " Save as" dialogue window
Undo	• returns the last performed step (the function is available only in programs with 2D environment and must be allowed in dialogue window "Options ")
Redo	• restores one returned step (the function is available only in programs with 2D environment and must be allowed in dialogue window "Options ")
Print and export document	• opens the dialogue window to create, edit and print output documents
Print and export picture	• opens the dialogue window to create, edit and print the current drawing displayed on the desktop
Copy	• copies the current picture displayed on the desktop or the inputted soil profile into clipboard
Insert	• inserts the soil profile stored in the clipboard (this option allows the user to copy the soil profile created in a different GEO program – e.g., from program "Pressures" to program "Gravity wall"

Tool bar "Scale and shift"

The tool bar buttons serve to manage all plots displayed on the desktop (zoom in/out, move...). The following figure shows locations of individual buttons:

Tool bar "Scale and shift"

Scale up	• scales up desktop view while keeping location of the point under the axis cross unchanged – this action is repeated using the left mouse button, the right button leaves the zooming mode
Scale down	• scales down the desktop view while keeping location of the point under the axis cross unchanged – this action is repeated using the left mouse button, the right mouse button leaves the zooming mode
Shows marked region	• shows and scales up the marked region - the region is selected using the left mouse button
Move displayed region	• moves the current view in an arbitrary direction – to proceed move mouse in the desired location while keeping the left mouse button pressed
Scale up	• scales up the displayed region while keeping the region centered
Scale down	• scales down the displayed region while keeping the region centered
Modify scale	• scales the view such that all objects are visible

Use previous scale
- modifies view scale - returns the original view scale prior to the last applied scale action

Tool bar "Plot setting"

The button on a tool bar serves to set the setting visualization style on the desktop (colors, thickness and style of lines, background….).

Toll bar "Plot setting"

Plot setting
- The button opens the "Setting visualization style" dialogue window that allows for setting all parameters of the picture displayed on the desktop.

Tool bar "Stage of construction"

Tool bar buttons serve to work with stages of construction. Picture shows location of individual buttons:

Tool bar "Stage of construction"

Adds construction stage
- adds new stage of construction at the end of list

Removes construction stage
- removes the last stage of construction from the list

[1] [2] Construction stage 1,2 ...
- switches between individual stages of construction – selection is performed using the left mouse button

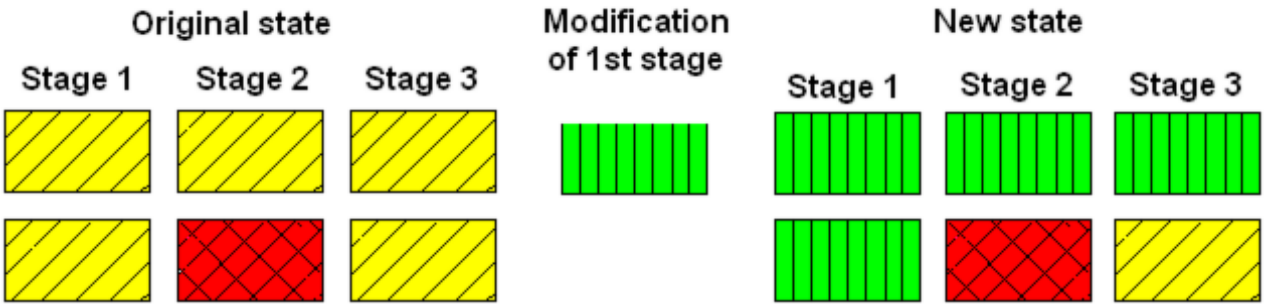
In all programs GEO5 this bar allows for defining stages of construction. Construction stages serve to model gradual building of a construction (essential for programs Sheeting verification, Settlement, FEM). This function can be also used for parametric studies and in each stage of construction assume different soil assignment or different design coefficients. It is rather advantageous to model earthquake effects on a structure in a separate stage of construction as it is then possible to assume different factors of safety or different design coefficients.
For individual types of input (soil assignment, anchors, supports…) there always exists relationship over construction stages. There are two types:

Defined heredity – (anchors, supports, surcharges…) – these objects always remember the stage, in which they were created. This is also the stage where these objects can be edited. In all subsequent stages these objects can be either removed or it is possible to change some of their properties (post-stressing anchor, changing surcharge magnitude, translating support…). When defining a new construction stage these objects are automatically carried over to that stage.

Automatic heredity – (assigning soil, terrain profile, water impact, analysis setting…) – for such types of inputs the properties from the previous stage are carried over to a new one if created. When changing properties in the current construction stage the program proceeds as follows:

- if the property in the next stage remains the same as in the previous stage it also receives the tag new – this change also applies to all subsequent stages

- if the property in the next stage differs from the one in the previous stage (this means that this property has been in the next stage already changed) then this change is not carried over to subsequent stages



Changes within stages of construction – automatic heredity

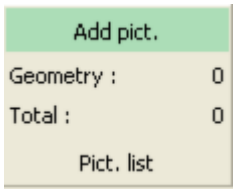
Vertical tool bars

The vertical tool bars serve to select the desired mode (regime) of inputting data (project, geometry, profile….) including analysis type and verification. Selecting the mode from this bar displays in the bottom part of the desktop the corresponding frame for data input.

Tool bar for switching between input data regimes

Standalone vertical tool bar serves to manage pictures.

The "Add picture" button opens the "New picture" dialogue window. The next line in the bar provides the number of stored pictures in a given regime of data input. The "Total" line shows the total number of stored pictures. The "Picture list" button opens the list of pictures .



Tool bar for controlling view manager editor

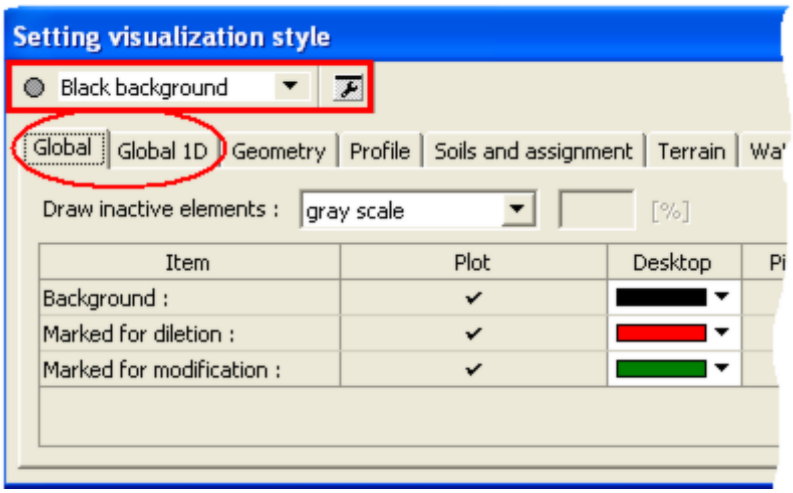
Setting visualization style

The "Setting visualization style" dialogue window serves to set the plot style (line type and color) for visualization on the desktop or for printout, respectively. It contains a group of tab sheets that correspond to individual data input regimes. The tab sheets serve to set the style

for drawing objects, which are specified in the related input regime.

The "Global" tab sheet defines the settings common to all input regimes (background color, color of elements to be deleted or modification and style of drawing of inactive elements).

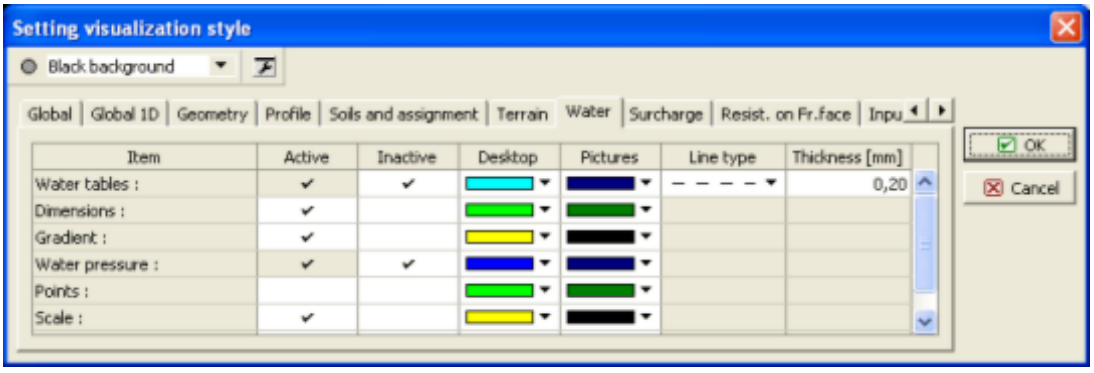
The program implicitly contains two standard settings of styles and colors, particularly for black or white background. The setting can be modified in combo list on the tool bar. User setting can be defined, i.e. the user can specify its own style of drawing and store that style with the help of button on the tool bar into the "Style manager".



Dialogue window "Setting visualization style" – global setting

The following picture shows an example of a tab sheet for setting the plot in the regime "Water". Individual columns of the table contain (moving from the left):

Item	• list of items plotted in a given input rgime (here, e.g., water tables, dimensions, gradient, water pressure....)
Active	• shows / hides a given item in the active regime "Water". In case the option cannot be turned off (the field has a gray base – in this case items "Tables" and "Water pressure") the visualization on the desktop is mandatory!
Inactive	• shows / hides a given item in other input data regimes. Visualization color depends on the assumed setting in the tab sheet "Global"
Desktop	• determines the item color displayed on the desktop
Pictures	• determines the item color displayed in the "Picture list" or on printout ("Print and export picture", "Print and export document")
Line type	• determines the line style
Thickness	• determines the line thickness



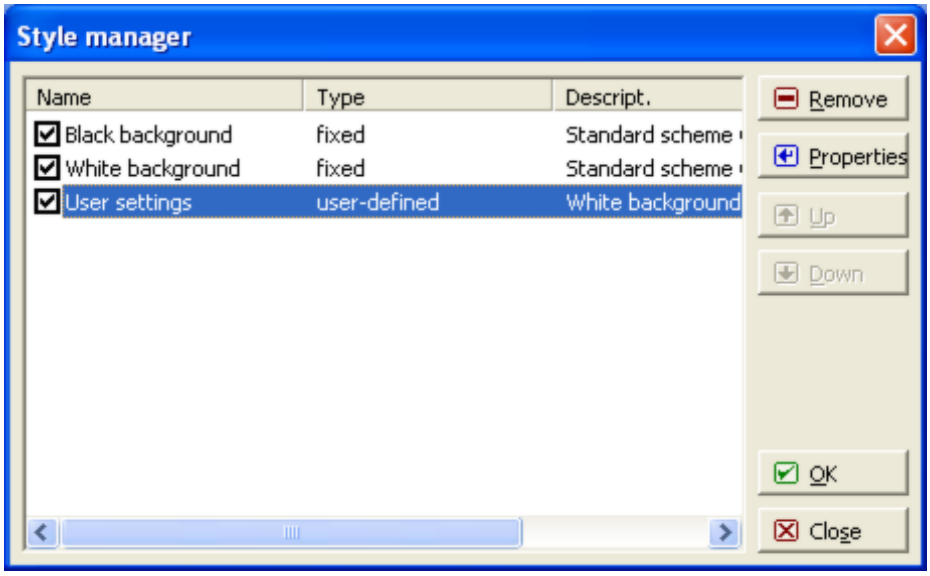
Dialogue window "Setting visualization style" – setting for the input regime "Water"

Style manager

The red button on a tool bar of the "Setting visualization style" dialogue window opens the "New style" dialogue window. The window allows for setting the style name and its description. The "OK" button saves the selected style.

Saving the user profile of visualization style

In such a way an arbitrary number of user profiles of visualization styles can be defined. The list of such profiles can be accessed from a combo list already containing implicitly predefined profiles (black and white background), or a view manager (can be opened by pressing the button on a tool bar) that allows for editing the profile. The buttons "Up" and "Down" serve to move within a list of the user defined profiles.

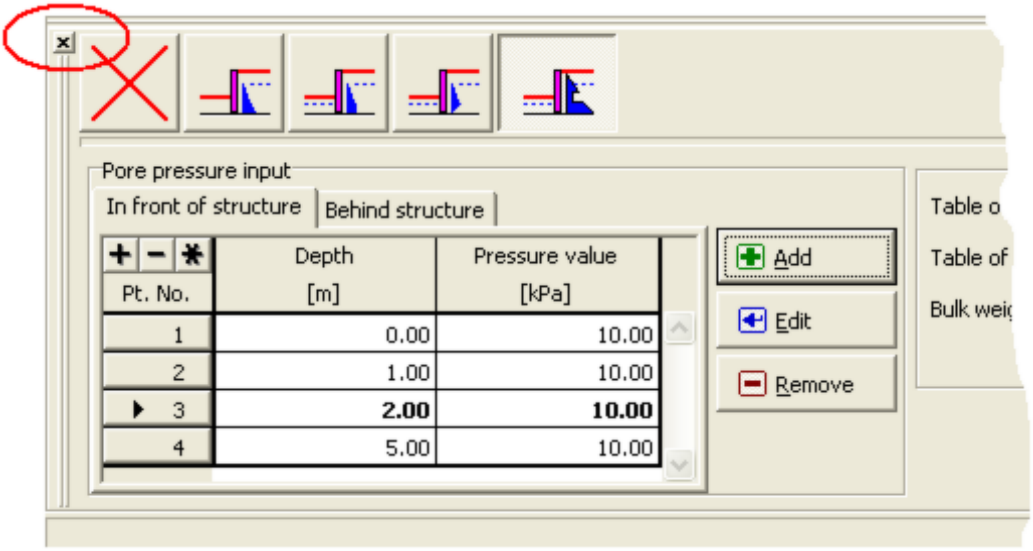


Dialogue window "Style manager"

Frames

The frame is a permanently opened window in the bottom part of the application window. Frames are changed depending on the input data regime of a given task selected from the vertical control bar. The frame may contain the following items: table, combo list, fields for inputting data (h1, h2....) and command buttons.

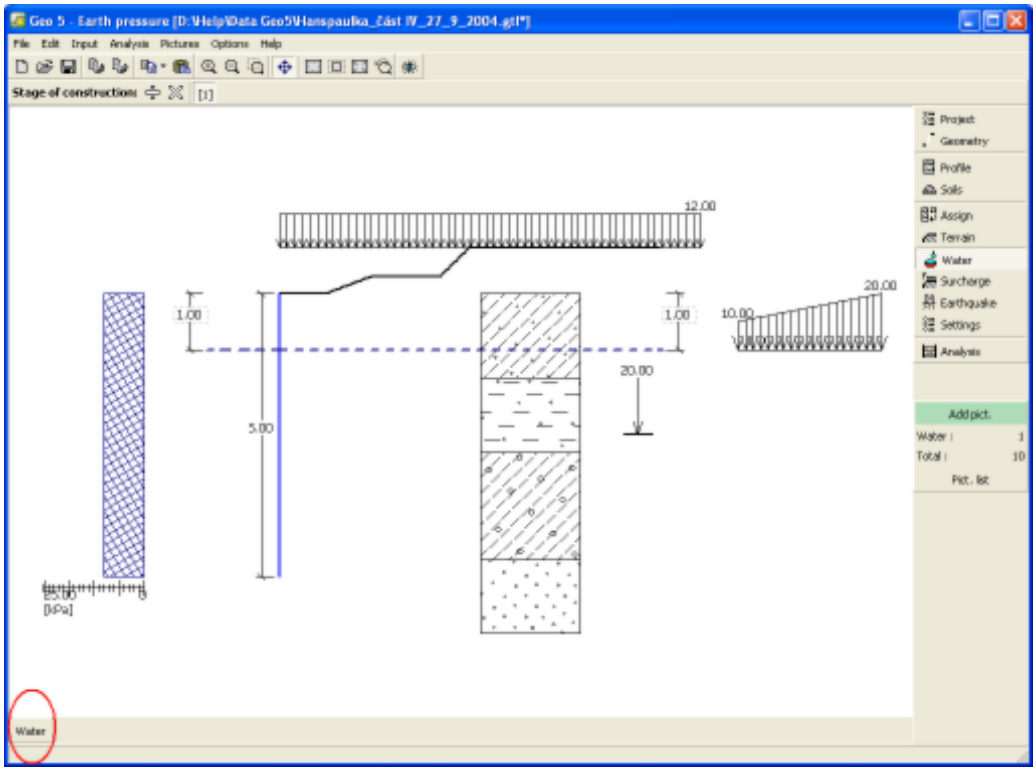
The function key "Tab" together with cursor arrows for moving within the selected element (e.g., combo list) and in case of command buttons the corresponding underlined letter ("Add" – "A") are employed when selecting the data using keyboard.



Frame control elements

The frame can be minimized using the button in the left upper corner. In this case the frame space is taken by the drawing space. In some cases it is more advantageous to exploit the frame space for increasing the drawing space, which is possible owing to the fact that the program uses the system of active dimensions and active objects so that the frame does not have to be displayed all the time.

Returning the frame to its original face is performed by pressing the button in left bottom corner of the desktop showing the frame name. Providing the frame is minimized, e.g. in the regime "Soils" it remains hidden even when switching to other input data regime.



Frame control elements

Tables

The table is a list of inputted data (for example a list of surcharges, soils, profile interfaces...). The table header contains a list of items (surcharge, name, width, size...) and in the upper left corner control elements to manage the table rows:

"+" • selects all table rows

"-" • cancels selection

"*" • inverts selection

The assumed selection can be also changed by pressing the desired row number. Buttons with numbers are "pressed".

The "Add" button opens the corresponding dialogue window for inserting the table data. If the list of data in the table is empty then all input fields in the dialogue window are empty. If the table contains some already inputted elements then the input fields are filled with values taken from the current table row (an "arrow" is positioned next to the row number). Elements (rows) are inserted in the table by pressing the "Add" button in the dialogue window. New data are placed at the end of the table.

Individual rows are edited with the help of the "Edit" button. Only the row marked with an arrow (see figure) will be edited regardless of other selected rows in the table. Some of the dialogue windows allows for editing a group of selected items using the "Edit selected" button. It is therefore possible to modify values in more rows all at once. Always the selected rows are edited.

The "Remove" button deletes all selected ("pressed") rows. More than one row can be removed at the same time. If not item is selected the program deletes the current row (marked with an "arrow").

It there is among rows selected for deletion a row, which cannot be deleted (e.g. starting point of a structure), the program stops the deletion process!

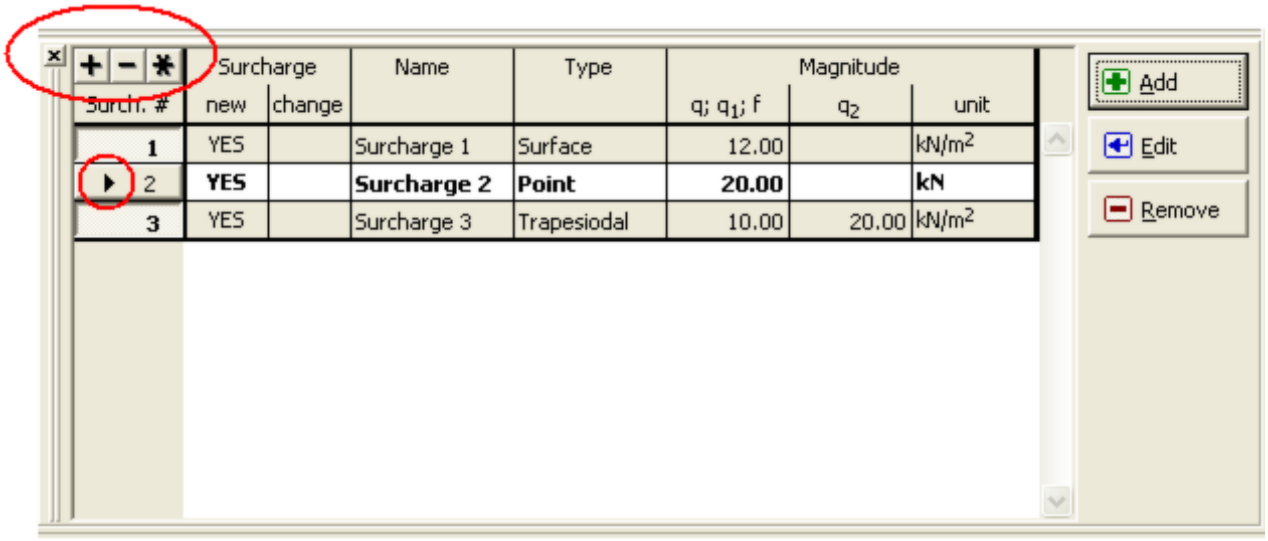
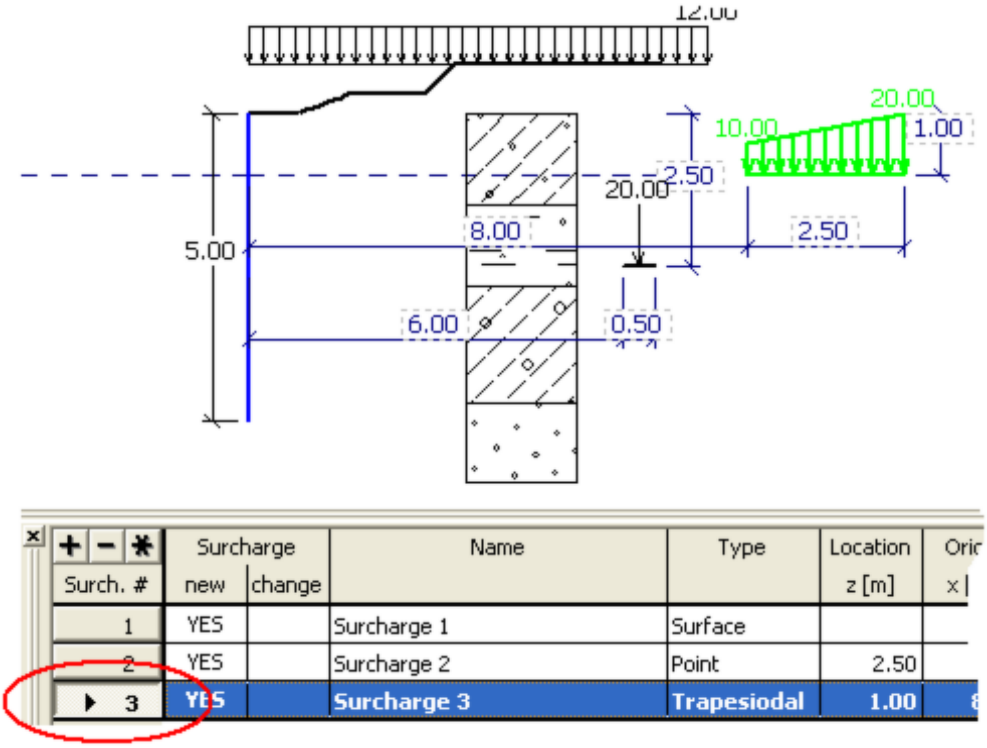


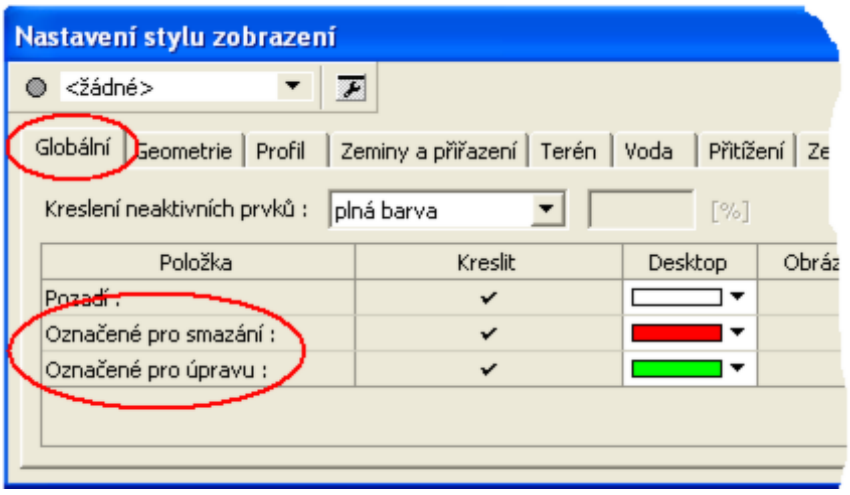
Table example

Selection state of individual table rows corresponds to visualization state of objects on the desktop. An object on the desktop that corresponds to the current row in the table (an "arrow" is positioned next to the row number) is implicitly displayed extra bold. If the row is selected ("pressed") the corresponding object is displayed in green. Pressing the "Remove" button colors all objects selected for deletion red.



Visualization of selected objects

Marking objects using these colors is implicitly set. This setting, however, can be modified in the "Setting visualization style" - tab sheet "Global".



Setting color for selecting objects

Dialogue windows

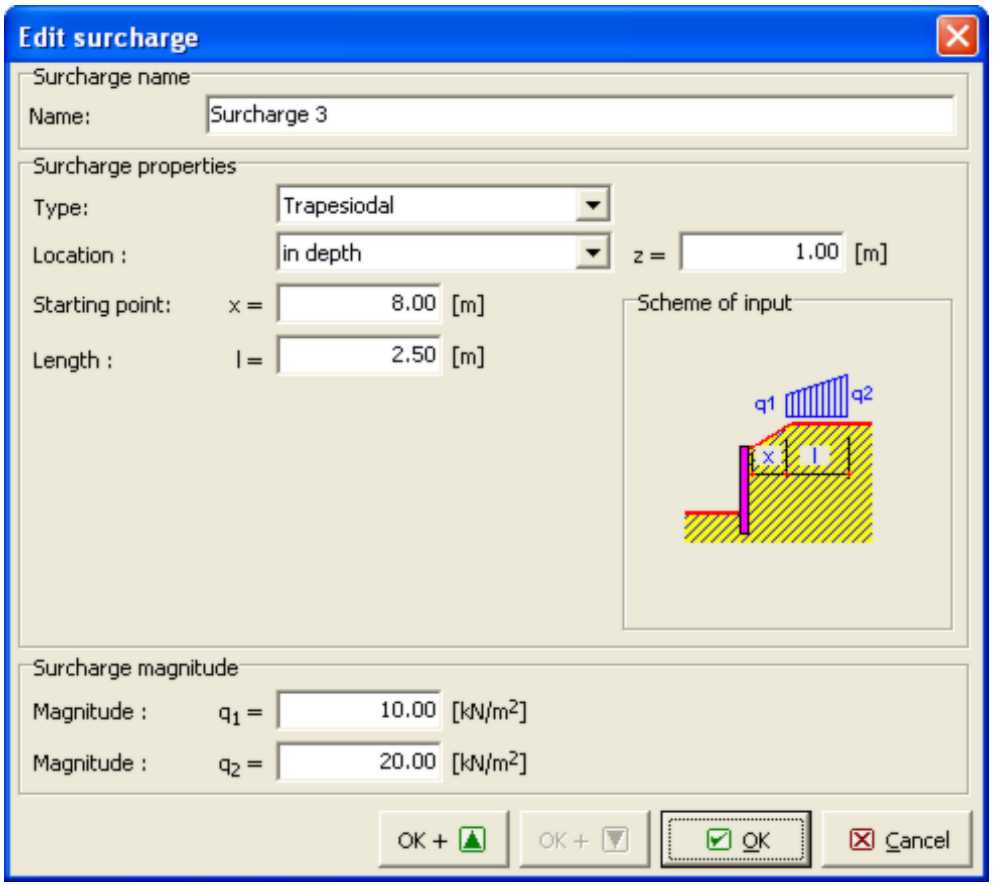
A dialogue window is one of the elements that allows for inputting data into program. In all GEO programs the dialogue windows apply to conventional windows management typical for the WINDOWS environment. A left mouse button is used when selecting objects in the window or alternatively the function key "Tab" when using keyboard. The cursor arrows, key "ENTER" or in case of command buttons the corresponding underlined letter ("Cancel" - "C", "OK" – "O") are employed when moving within an object.

A dialogue window can contain the following items: table, combo list, fields for inputting data (number, text) and command buttons. The "OK" command button confirms the selection, while the "Cancel" button leaves the input mode.

Providing the window contains a certain non-typical control element (or this element has

some other rather then typical effect) its function is described in the corresponding data input regime.

As an example consider the following picture showing the "Edit surcharge" dialogue window that contains the "OK + ↑" and "OK + ↓" buttons. These buttons allow the user to move within the list of inputted surcharges and at the same time to confirm changes made in the window. Pressing this button results in the same action as if closing the window with the "OK" button and opening it again for the next element in the list.



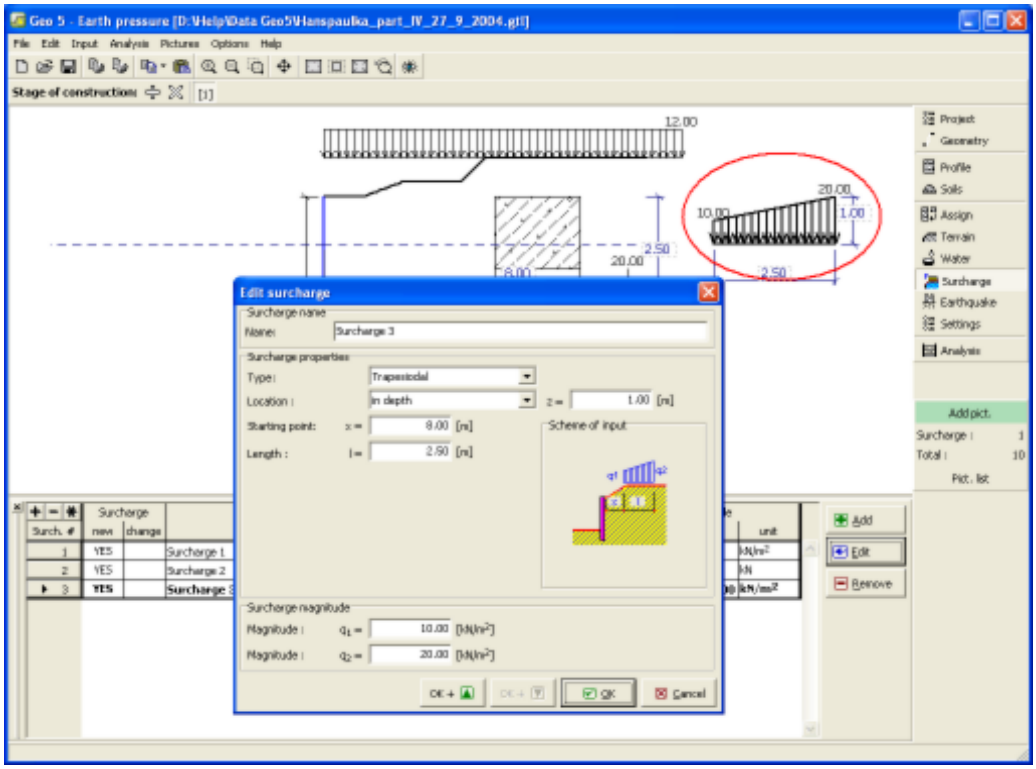
Dialogue window example

Active dimensions and objects

The system of active dimensions and objects allows for faster editing of the inputted data.

Active dimension is a dimension that can be edited directly on the desktop. The value of active dimension is labeled by frame (dashed line). Positioning the mouse cursor and the frame changes its mask into a "hand". Clicking the value than changes the frame view (is plotted in a solid line), to cursor starts to blink and the dimension can be edited. The "Enter" button closes the editing mode. The change is immediately displayed on the desktop.

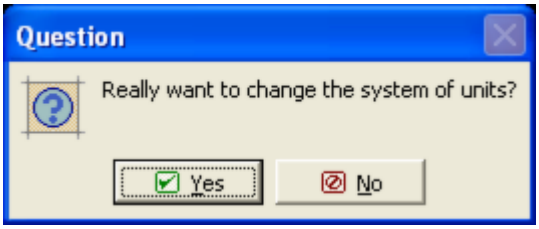
Active object functions in a similar way. Changing the cursor mask into a "hand" and clicking the object (double click) than activates the editing mode. In this case, however, the values are not edited directly on the desktop, but rather in the dialogue window originally used to create the object. The picture shows an example of an active object (trapezoidal surcharge), when clicking on the desktop opens the "Edit surcharge" dialogue window.



Example of using active dimensions and active objects

Unit - metric / imperial

The program allows for choosing either metric or imperial units. Use combo list to select the desired type of units. A prompt message appears requesting to confirm the selection.



Dialogue window to confirm the change of units

Copy to clipboard

The program allows for using the Windows clipboard in two different ways:

- it is possible to copy the current desktop view. The picture can be then inserted into an arbitrary editor (MS Word, Drawing, Adobe Photoshop, etc.) Individual parameters are set in the "Options" dialogue window, tab sheet " Copy to clipboard"
- it is possible to copy the program input data (soil parameters, profile and interfaces, surcharges, water impact, terrain, etc.). The copied data can be then inserted into another GEO5 program

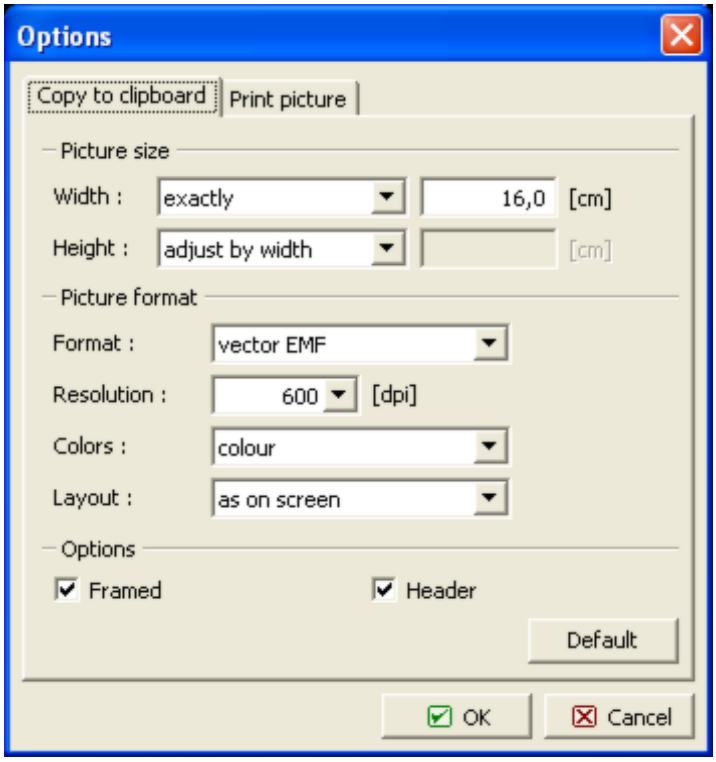
Copying to clipboard is available either from the control menu (items "Modifications", "Copy picture") or using the "Files" button on the tool bar.

Options

The "Options" dialogue window serves to set some of the special program functions (copy to clipboard, print view, grid and step, etc.).

This dialogue window is opened from a control menu (items "Setting", "Options").

The window contains individual tab sheets (number and content may vary depending on individual programs) that allow for specifying corresponding settings.



Dialogue window "Options"

Options - copy to clipboard

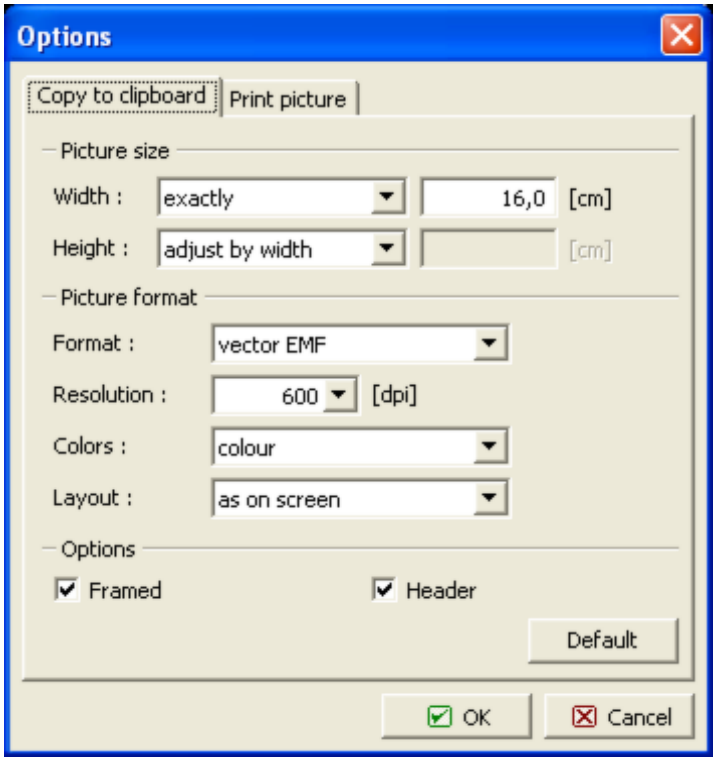
The "Copy to clipboard" tab sheet allows for setting the controlling parameters:

Picture size
- the setting defines the picture size. Either the picture width or height can be assigned manually. The other dimension is always set automatically

Picture format
- the setting defines the picture format (*.EMF, *.WMF, *.BMP), its resolution and color. Recommended setting is displayed in the figure (format: *.EMF, resolution: 600 dpi, color)

Options
- the setting defines the picture frame and header. If both options are checked, the picture contains both the frame and header

The "Default" button in the window sets original implicit values.



Dialogue window "Options" – tab sheet "Copy to clipboard"

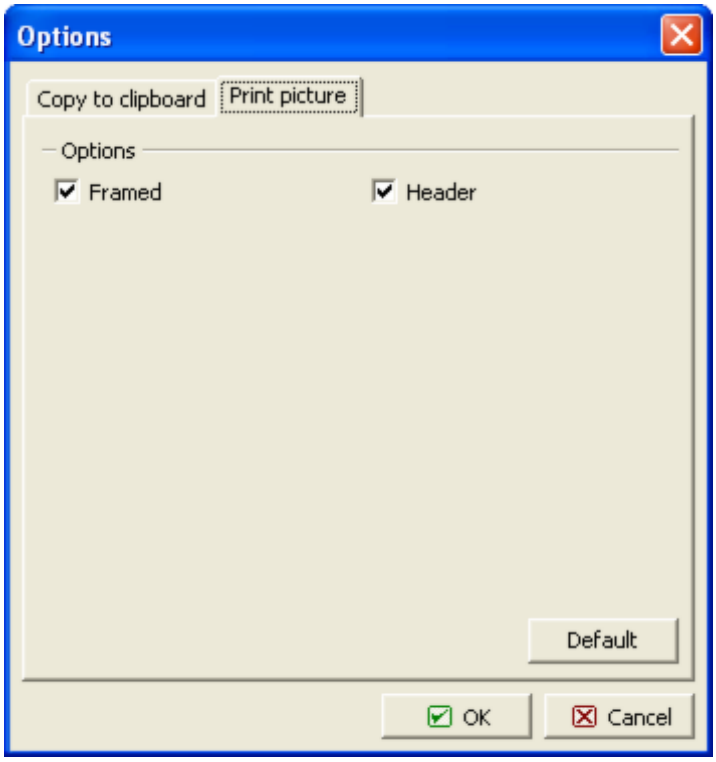
Options - print picture

This dialogue window is opened from the control menu (items "Setting", "Options"). The "Print picture" tab sheet allows for setting the picture parameters assumed for printout or export in the "Print and export picture" dialogue window.

Options

- the setting defines the picture frame and header. If both options are checked, the picture contains both the frame and header

The "Default" button in the window sets original implicit values.



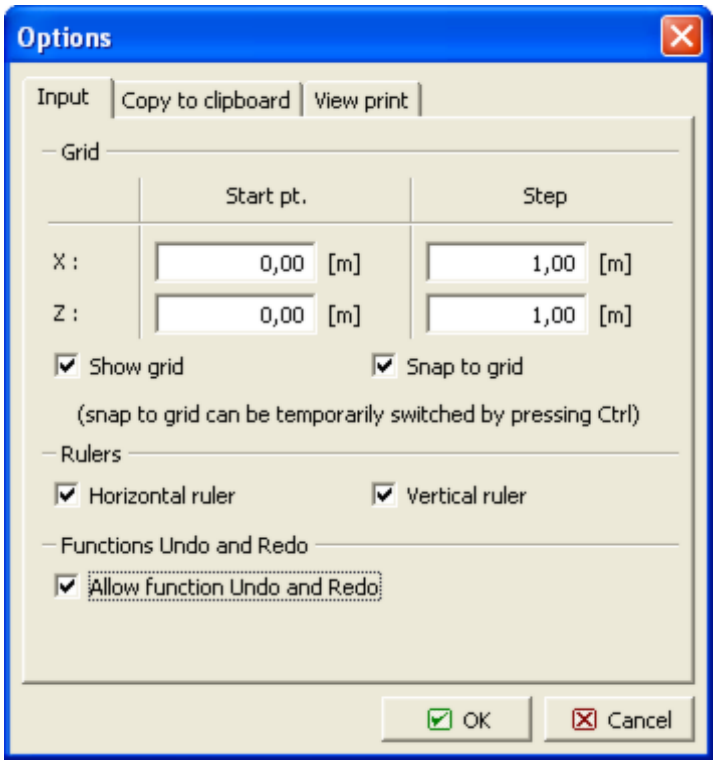
Dialogue window "Options" – tab sheet "Print picture"

Options - input

The "Options" dialogue window, tab sheet "Input" allows for setting the "Grid" parameters and parameters of functions "Back" and "Repeat".

This tab sheet is implemented only in 2D programs (Slope stability, Settlement, Beam, etc.).

Grid	• sets the grid origin and step in the X and Z directions
Show grid	• shows / hides grid on the desktop
Snap to grid	• turns on / off the snap to grid option using mouse (when shifting the mouse the cursor jumps over the defined grid – a point off the grid can be specified by holding the "CTRL" key)
Horizontal rule	• shows / hides horizontal rule with a scale of distances on the desktop
Vertical rule	• shows / hides vertical rule with a scale of distances on the desktop
Functions Back and Repeat	• turns on / off the possibility of using these functions in the program (on horizontal tool bar these buttons are "foggy"



Dialogue window "Options" - tab sheeet "Input"

Common input

This chapter contains the hint sections that either provide details on the chapter "Data input and analysis regimes" or are common to several GEO programs.

Project - Earth pressures

The "Project" frame – tab sheet "Earth pressures" contains basic settings driving the analysis of earth pressures. The program offers presetting for various countries (Czech

Republic, Germany, France....) and "Standard setting" recommended by the authors for countries stored in the list.

When the "User" option is selected it is possible choose from an arbitrary type of analysis offered by the program:

Earth pressure analysis

Methods to compute active pressure

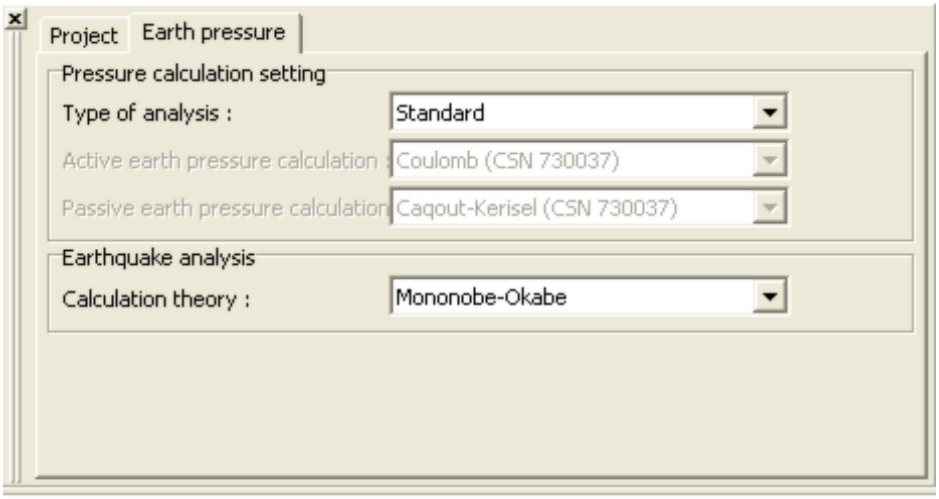
- Caqout-Kerisel
- Coulomb (CSN 730037)
- Müller-Breslau (DIN 4085)
- Mazindrani (Rankin)
- Absi

Methods to compute passive pressure

- Caqout-Kérisel (CSN 730037)
- Coulomb
- Müller-Breslau
- Sokolovski (DIN 4085)
- Mazindrani (Rankin)
- Absi

Earthquake analysis

- Mononobe-Okabe
- Arrango



Frame "Project" - tab sheet "Earth pressures"

Inputting and editing soils

The "Add new soil" dialogue window serves to input name soil parameters that should be obtained from laboratory measurements or geological survey.

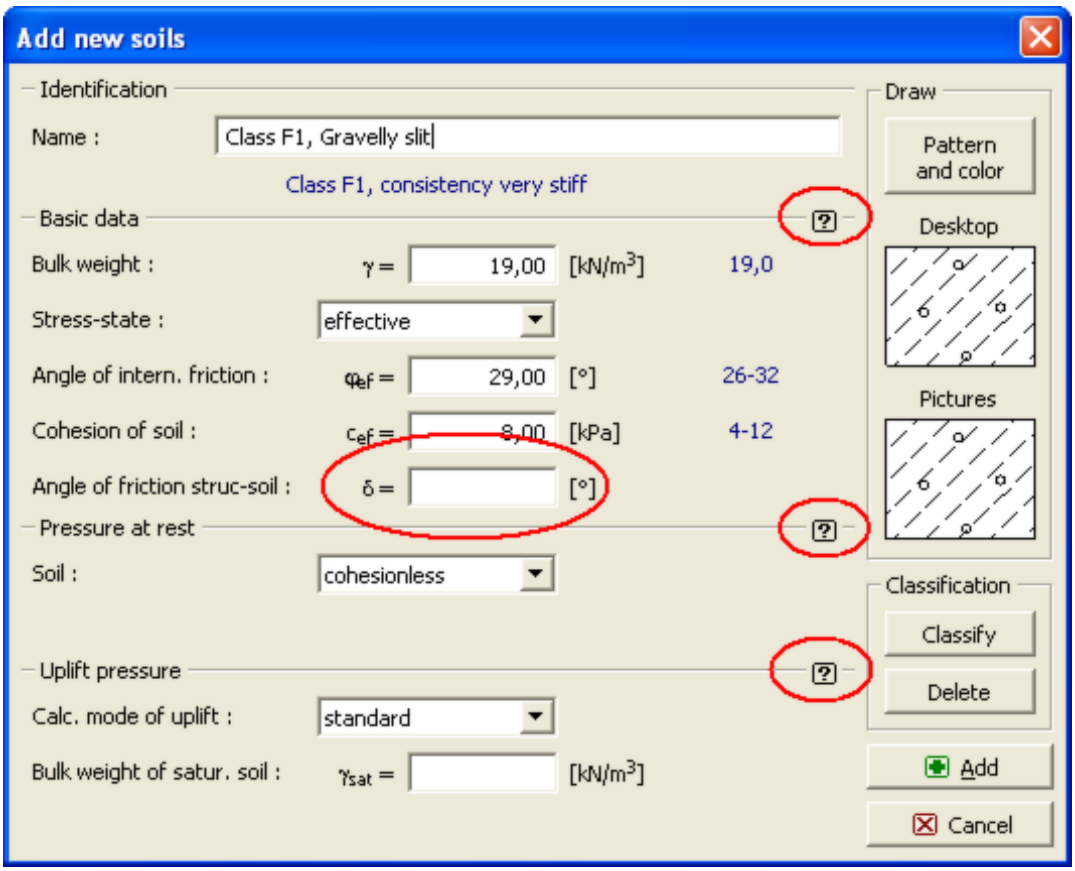
All inputting fields in the window must be filled. The only exception is the value of γ_{sat} (bulk weight of saturated soil) in the window section "Uplift". Should this filed remain empty the program autonatically adds the value of γ (bulk weight of soil).

Cliking the hint button " " provides information about the theory of analyses linked to individual values being inputted.

Color and soil sample are selected in "Soil and rock symbols" dialogue window. To open this window press the "Sample and color" button.

If no geological survey or laboratoray experiments are available, the soil can be specified with the help of soil database containg approximate values of basic characteristics. The "Classify" button opens the "Soil classification" dialogue window with the values offered for inserting into the window. The "Delete" button allows for removing the information about classified soil from the catalogue. Soil parameters that do not appear in the catalogue ("Friction angle struc-soil" in the picture) must be in any case assigned manually.

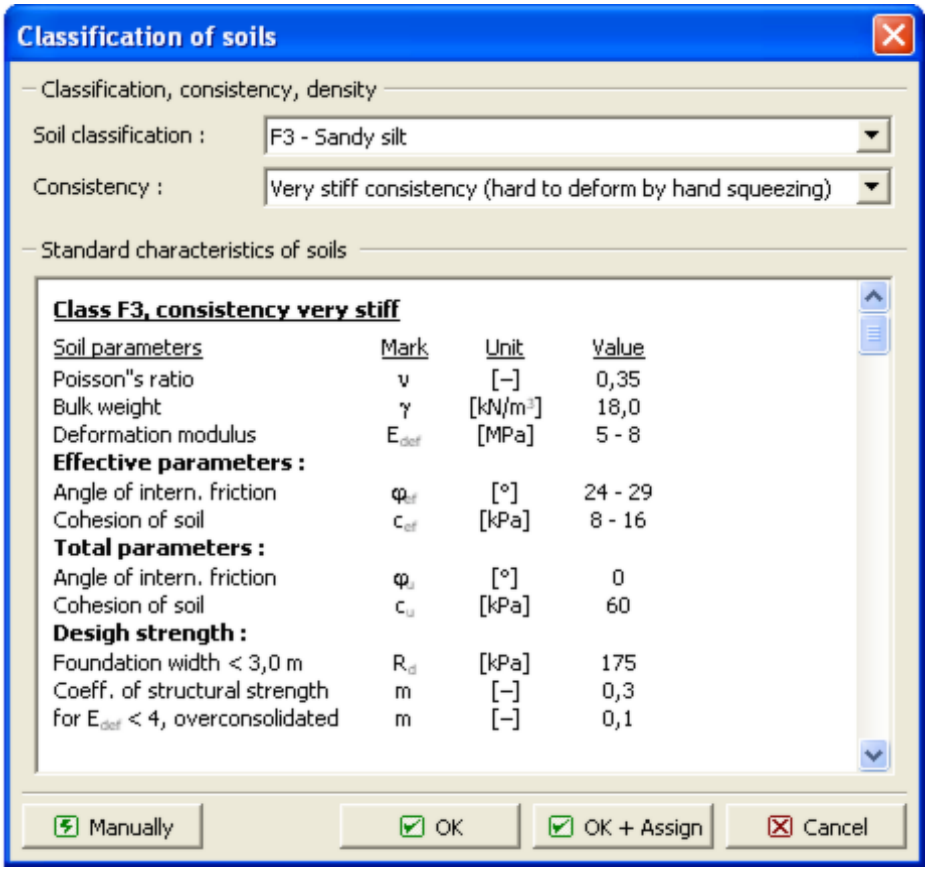
The specified soil is inserted into the list of soils by pressing the "Add" button.



Dialogue window "Adding new soils"

Soil classification

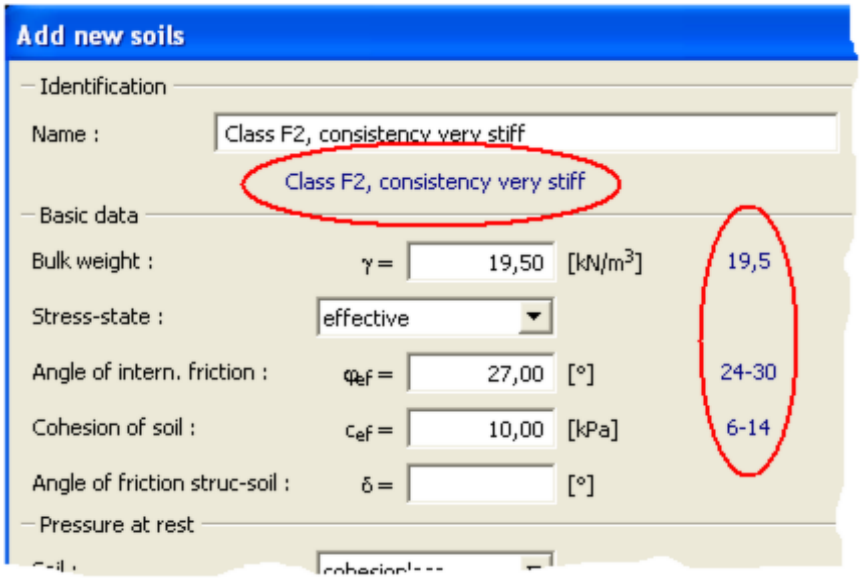
Approximate values of soils can be obtained from the catalogue of soils according to CSN 73 1001 standard "Foundation soil below spread footings". The combo list serves to select the desired soil and specify its consistency or compactness, respectively. The soil parameters obtained from the catalogue appear in the window.



Dialogue window "Soil classification"

Pressing the "OK" button shows in the "Add new soil" recommended values next to corresponding input fields (see Fig.). Pressing the "OK+Assign" button then assigns to individual input fields the average values of soil parameters. The "Cancel" button leaves the window with no action.

The "Manual" button opens the "Manual soil classification" dialogue window that allows for classifying the soil if its parameters are known, e.g., from laboratory measurements (grading, moisture, compactness....).



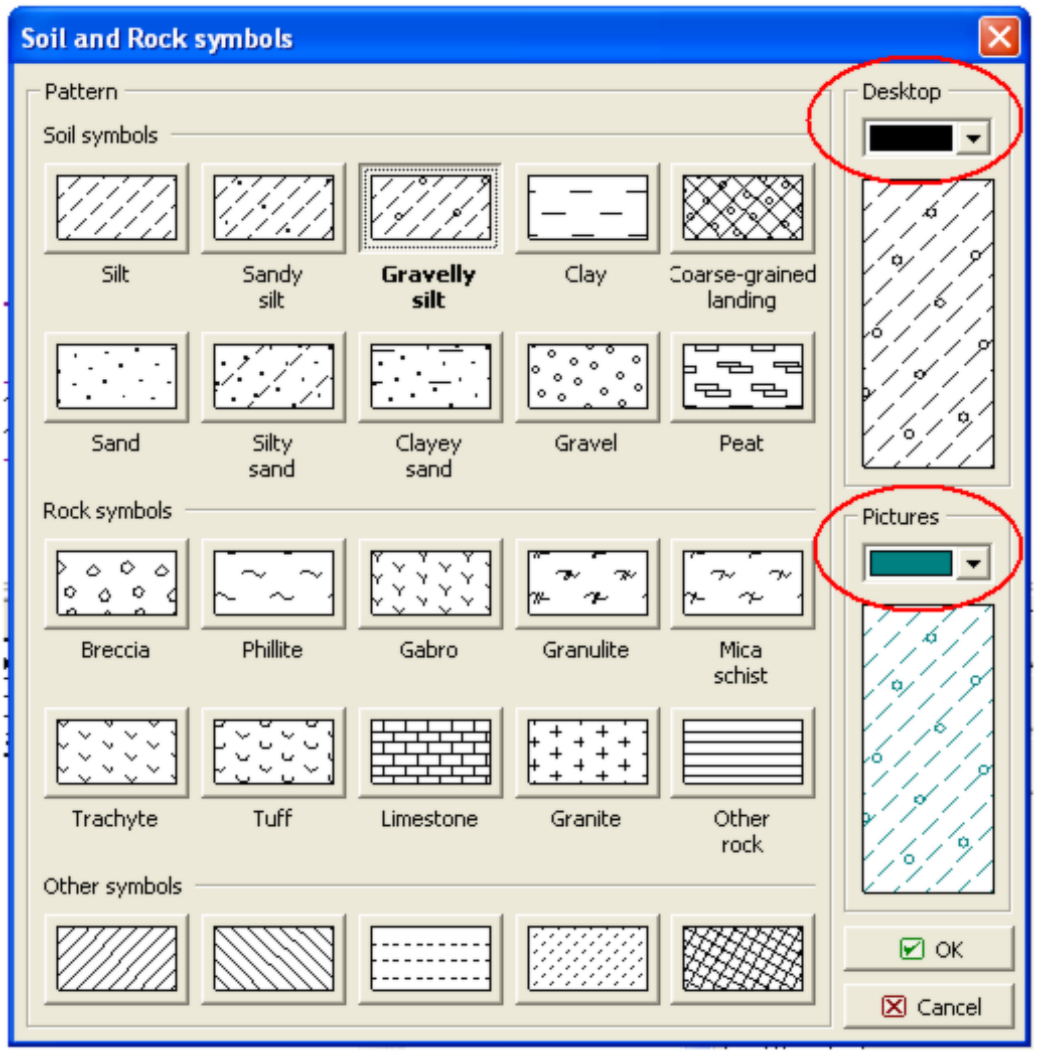
Soil classification – recommended range of values

Soil and rock labels

The soil inputted either manually or inserted from the catalogue of soils can be assigned a sample (type of hatching) and color to be shown in the profile on the desktop.

A color selected from the "Desktop" combo list is the color used to plot soils (rocks) on the desktop. A color selected from the "Pictures" combo list is the color used to visualize soils (rocks) in pictures that are either stored in the "Picture list" or printed with the help of "Print and export pictures".

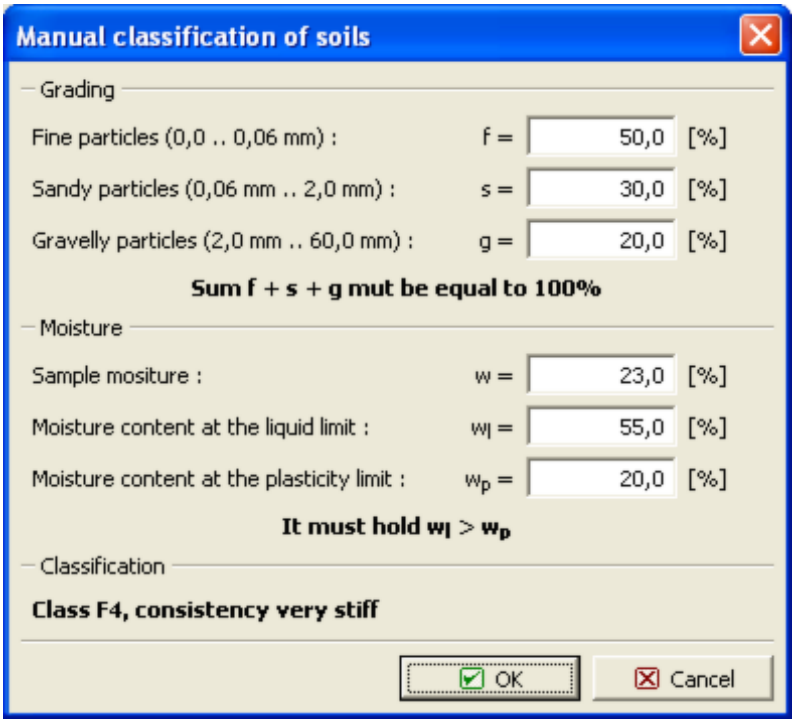
The sample color to be sufficiently visible should be chosen with respect to the desktop background or printout paper, respectively.



Dialogue window "Soil and rock symbols"

Manual classification of soil

This dialogue window allows for specifying the soil parameters, which serve to add the soil into a catalogue of soils. The "OK" button switches back to the "Soil classification" dialogue window with setting and classified soil.



Dialogue window "Manual classification of soil"

Interface in 2D environment

The left part of the desktop contains a table with the list of interfaces. Interfaces are ordered in the table from top to bottom. For currently selected interface the program displays in the mid section of the desktop another table listing individual interface points.

A tool bar in the top part of the desktop contains control buttons to manage interfaces.

Tool bar "Interface"

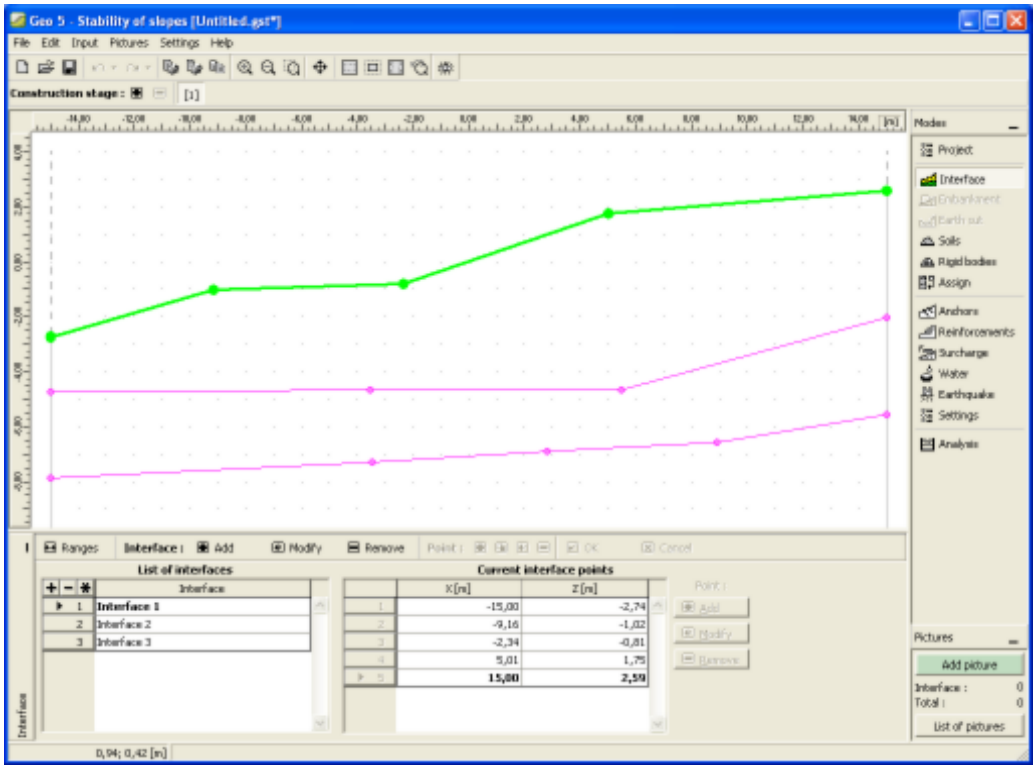
Margins
- opens the "World coordinates" dialogue window that allows for setting the world dimensions (left and right edge).

Add
- turns on the regime for inputting a new interface – individual interfaces can be added in an arbitrary order. Each interface is automatically stored in the list of interface when leaving the input mode

Modify
- turns on the regime for editing an interface – this regime is also activated by clicking the desired interface on the desktop

remove
- upon pressing the "Remove" button the program marks the selected interface with a red color and opens the dialogue window to confirm this action

Every change made to a given interface can be put back using the "UNDO and REDO" buttons on the horizontal tool bar.



Frame "Interface"

Adding interface

The "Add" button starts the regime for inputting points of a new interface. Other buttons on the same tool bar assumed for inputting and editing interface points become active. The "OK" button (tinged green) closes the input regime and stores the inputted points. The "Cancel" button (tinged red) closes the input regime without accounting for changes.

Two options are available to specify coordinates of individual interface points:

Using table: interface points are introduced in the "New interface points" table. The "Add" button opens the "New point" dialogue window that allows for specifying coordinates of a new point. The "Add" button then inserts the point into the table. The "Cancel" button is serves to close the input mode when all interface points are introduced. The "Edit" and "Remove" buttons allow for either editing or deleting the inputted points. Each change in interface geometry immediately appears on the desktop.

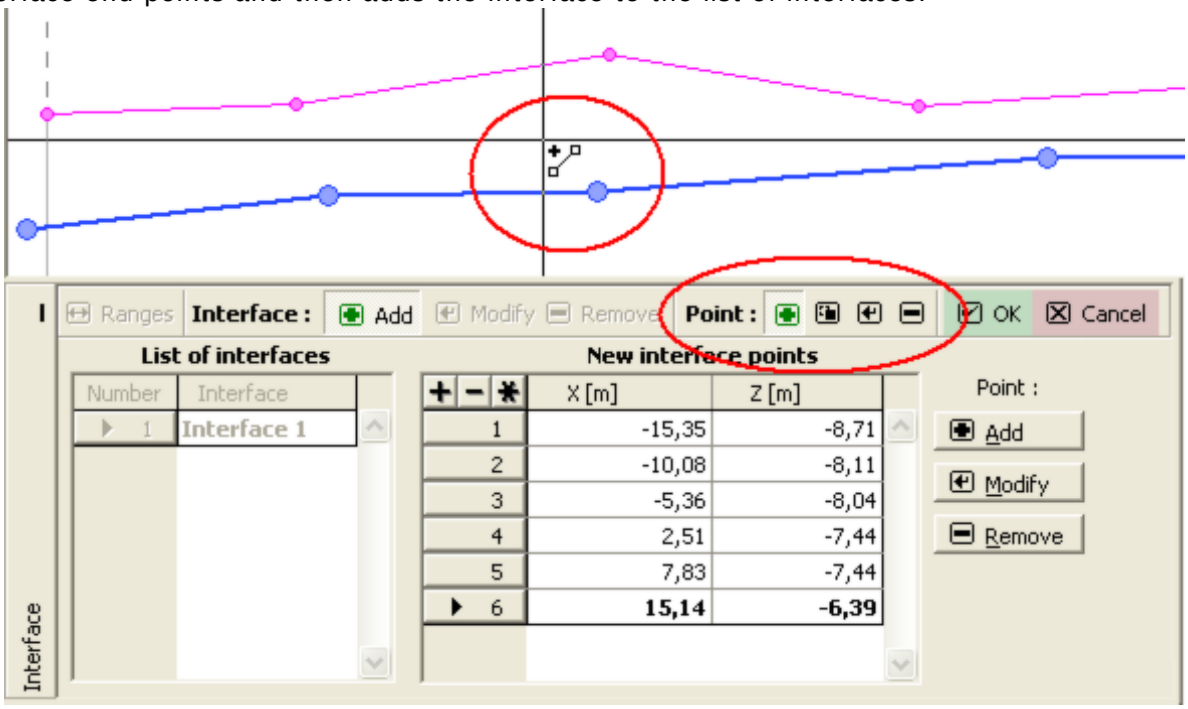
Using mouse: individual buttons on the vertical tool drive this inputting mode. The following modes are available:

Add point	• the point is inserted by clicking the left mouse button on the desktop - the grid option can be exploited when inserting the point - the inputted point is automatically rounded to two decimal digits – both mouse and keyboard input modes are therefore identical
Edit point using mouse	• pressing the existing interface point using the left mouse button allows for selecting this point and then moving it to a new position
Edit point in dialogue point	• clicking the existing interface point opens the dialogue window that allows for modifying the point coordinates
Remove point	• pressing the existing interface point using the left mouse button opens the "Remove point" dialogue window – when confirming this action the point is deleted

The program allows also for introducing vertical interfaces – in such a case the program requests to insert the point either to the left or to the right. The buttons that serve to confirm the action are colored – the same color is also used to visualize both input variants on the desktop.

The "OK" button (tinged green) is used to store the inputted interface when all interface points are introduced.

The program also contains an automatic corrector of inputted interface that determines the interface end points and then adds the interface to the list of interfaces.



Frame "Interface"

Editing interface

The "Modify" button turns on the regime of inputting points of a new interface. The program also opens the currently selected interface (it is selected from the "List of interfaces" table; it is displayed as a solid thick line on the desktop). An interface can be selected also as an active object by clicking a point, line or interface using the left mouse button.

The actual editing procedure (adding, shifting, deleting points) is the same as for adding interface input mode.

The "OK" button closes the editing regime and stores all performed changes. The "Cancel" button closes the regime without accounting for all previously made modifications.

After leaving the edit mode (similarly to adding interface), the program immediately runs the corrector of the inputted interface to check for the interface shape and if necessary to modify the interface end points.

Corrector of inputted interface

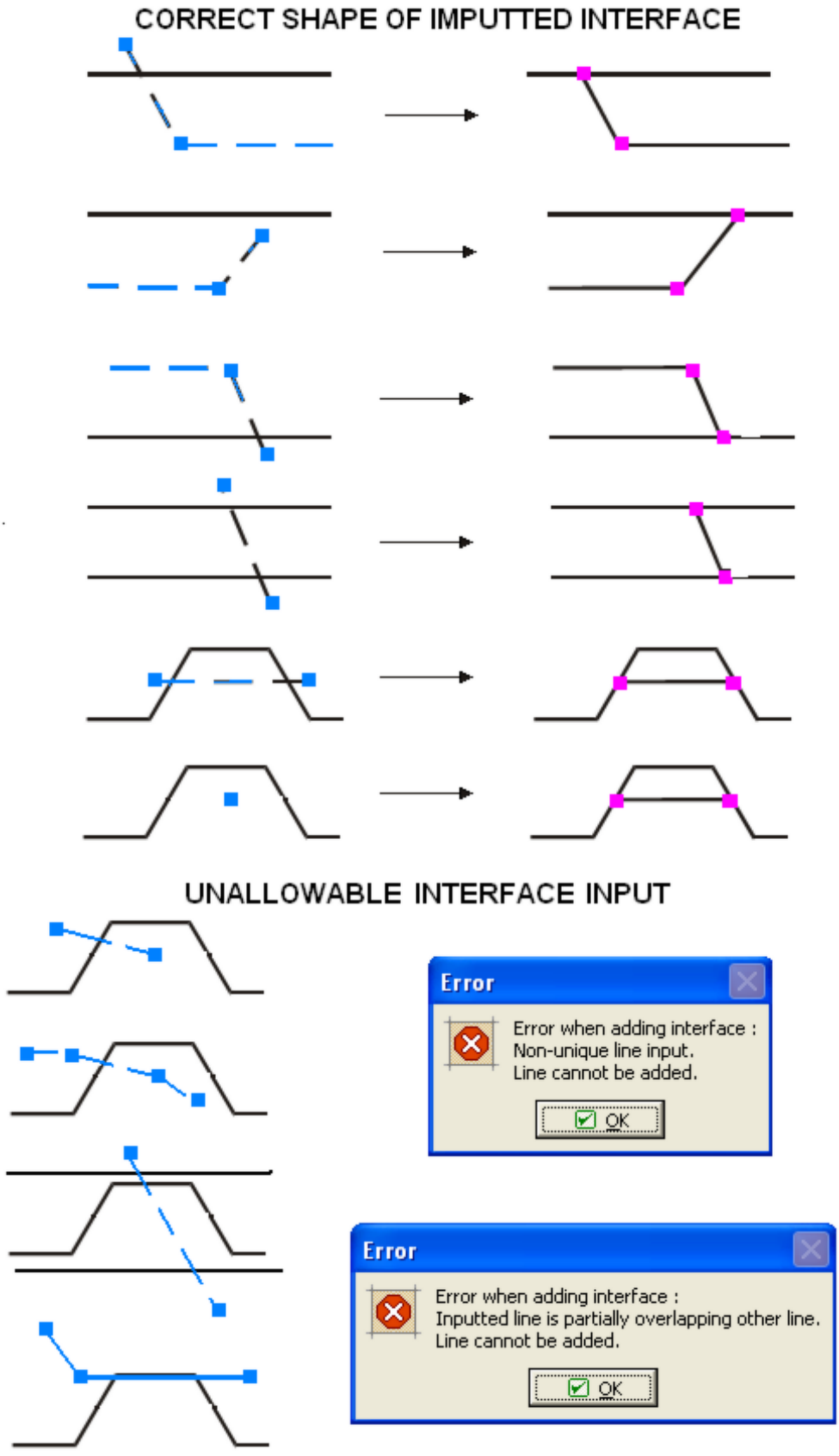
When the inputting or editing process is completed the program automatically modifies the inputted interface to comply with the program requirements, i.e. the end points touch the world edges or other interfaces. The automatic corrector can be further used to simplify the input process – e.g., if only one point is used to specify an interface the program automatically creates a horizontal interface containing the inputted point.

If the interface touches another interface the corrector creates new end points of the current interface. These points then also become the points of the interface being touched. All lines of individual interfaces thus start and end in a point.

In case of an incorrect input (see the picture below) the interface cannot be stored. In this case the interface must be modified or the inputting process must be stopped using the "

Cancel" button.

Here we present examples of interface corrector functions (correct and incorrect input):

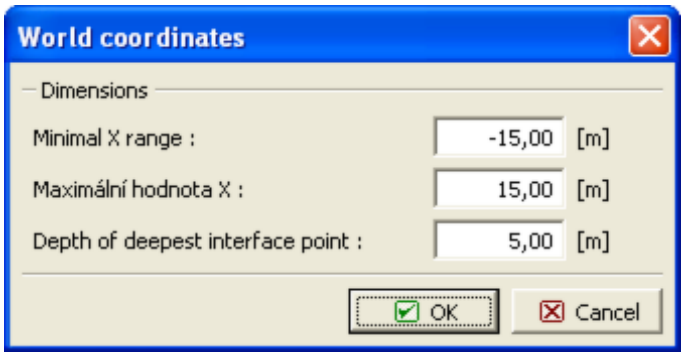


Correct and incorrect interface shapes

World coordinates

The dialogue window serves to specify world coordinates (dimensions) for a given task – left and right edges. The third data is auxiliary – it determines the depth of drawing earth profiles on the desktop – it has no effect on the performed analysis.

The world coordinates can be changed at any time – when increasing dimensions all inputted interfaces are automatically prolonged, when reducing dimensions all points falling off the new world coordinates are automatically removed.

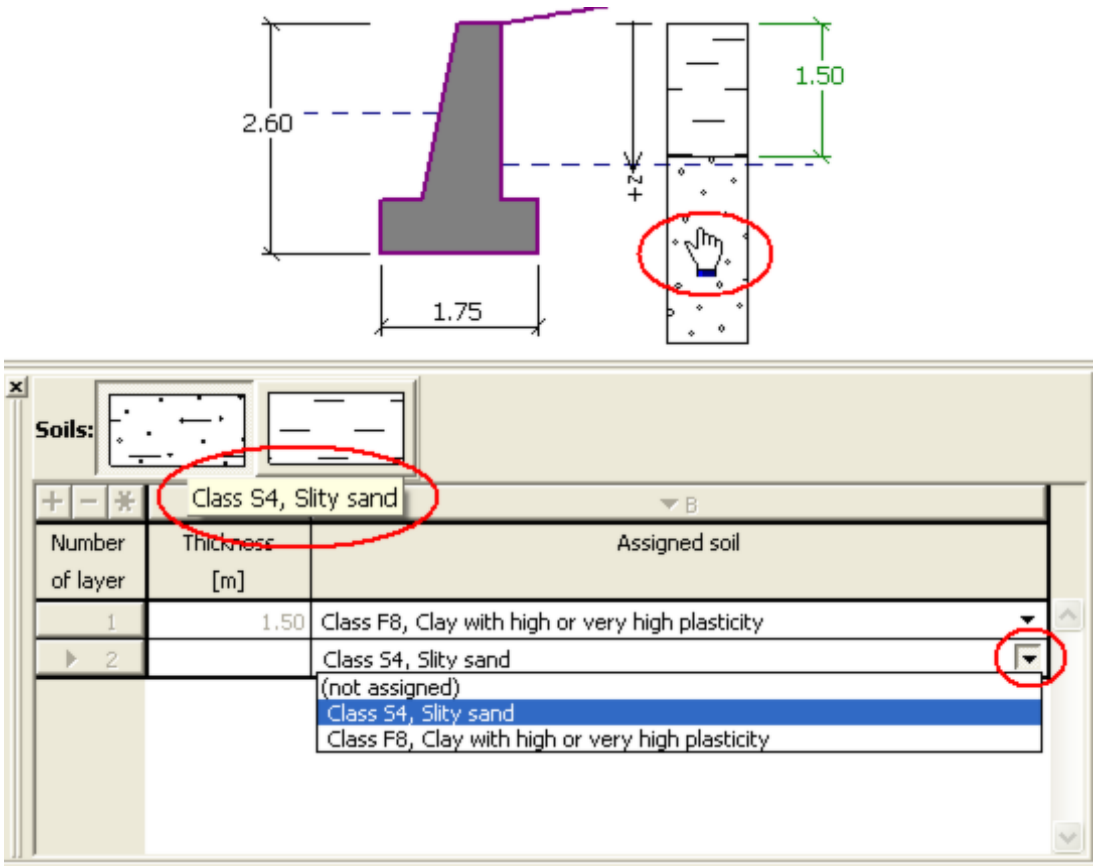


Dialogue window "World coordinates"

Assigning soils

Two options are available to assign soils into individual profile layers. Clicking the left muse button on the tool bar button above the table selects the desired soil (positioning the mouse cursors in the bar above the soil button displays a bubble hint with the soil name). The soil is inserted by moving the mouse cursor (the cursor mask changes into a "hand") first into a specific layer and then by pressing the left mouse button.

The second option requires opening a combo list of a specific interface and then selecting the desired soil to be assigned. All changes in the soil assignment are automatically displayed on the desktop.



Frame "Assigning"

Design coefficients

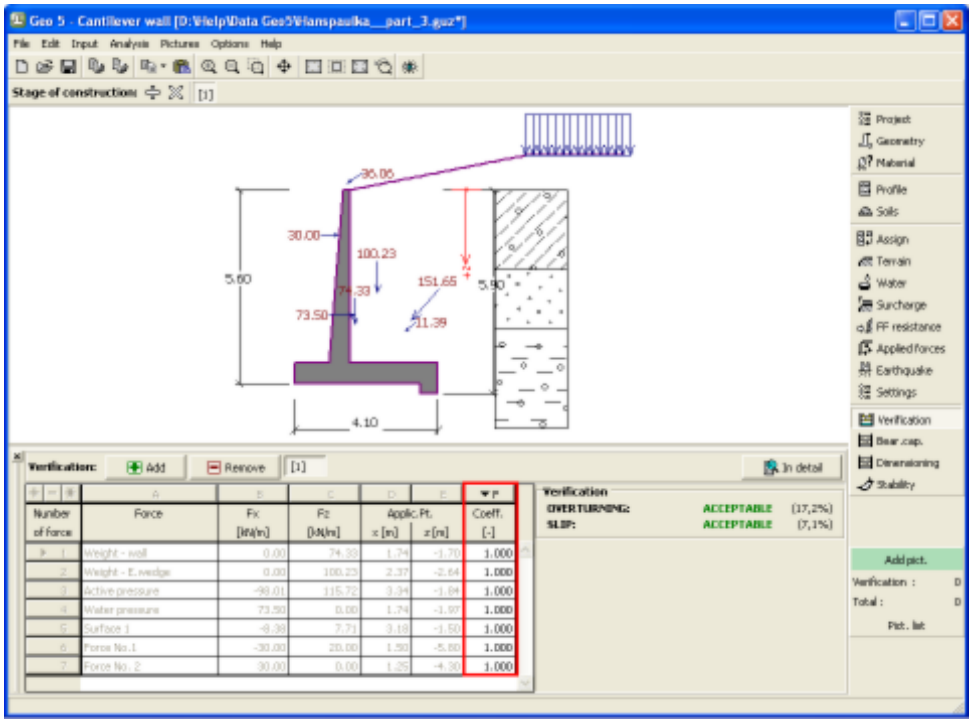
The "Analysis" or "Verification" frames that display the list of computed forces allow for specifying design coefficients. A design coefficient multiplies the corresponding force. When inputting the coefficient the results are automatically recomputed and the desktop shows modified forces.

Design coefficients are advantageous for example for:

- structure testing when a structure response to an increase of force specified directly in the analysis window can be visualized
- excluding several forces from verification or their reduction
- Specifying design combinations – e.g., different coefficients can be assigned in the sense of EC to main load variables and side variable loads

The following combinations can be for example specified when performing the wall verification:

	Analysis 1	Analysis 2	Analysis 3
• Wall	1,0	1,0	1,0
• Active pressure	1,0	1,0	1,0
• Surcharde 1	1,0	0,5	0,5
• Surcharde 2	0,5	1,0	0,5
• Surcharde 3	0,5	0,5	1,0



Frame "Verification" – application of design coefficients

Running more analyses / verification

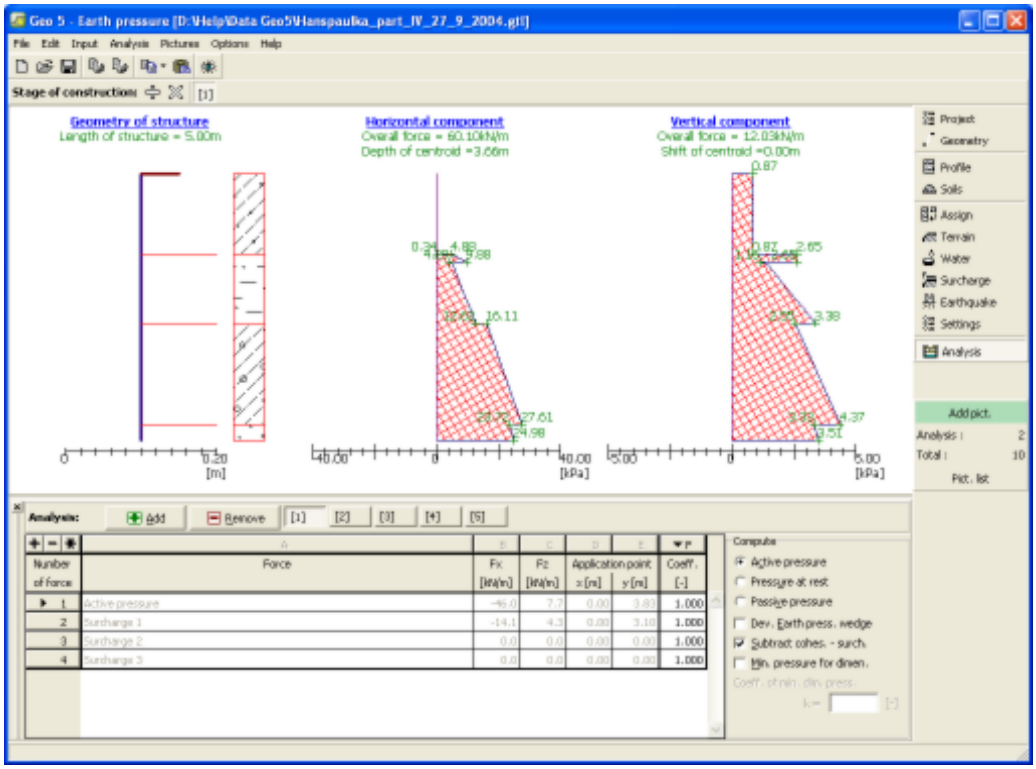
Most frames that display the analysis results allow for defining more than one analysis to be run. Several analyses within one construction stage are carried out for example for:

- dimensioning structure in more locations
- analyses of various slip surfaces
- verification with various design coefficients

The bar in the top part of the frame serves to manage individual analyses.

Frame "Analysis" – tool bar "Running more analyses / verification"

Add	Add	• adds additional analysis on the bar
Remove	Remove	• removes the currently selected analysis
[1] [2]	Analysis 1,2,3 ...	• switches between individual analyses



Frame "Analysis" - "running more analyses / verification"

Connecting programs

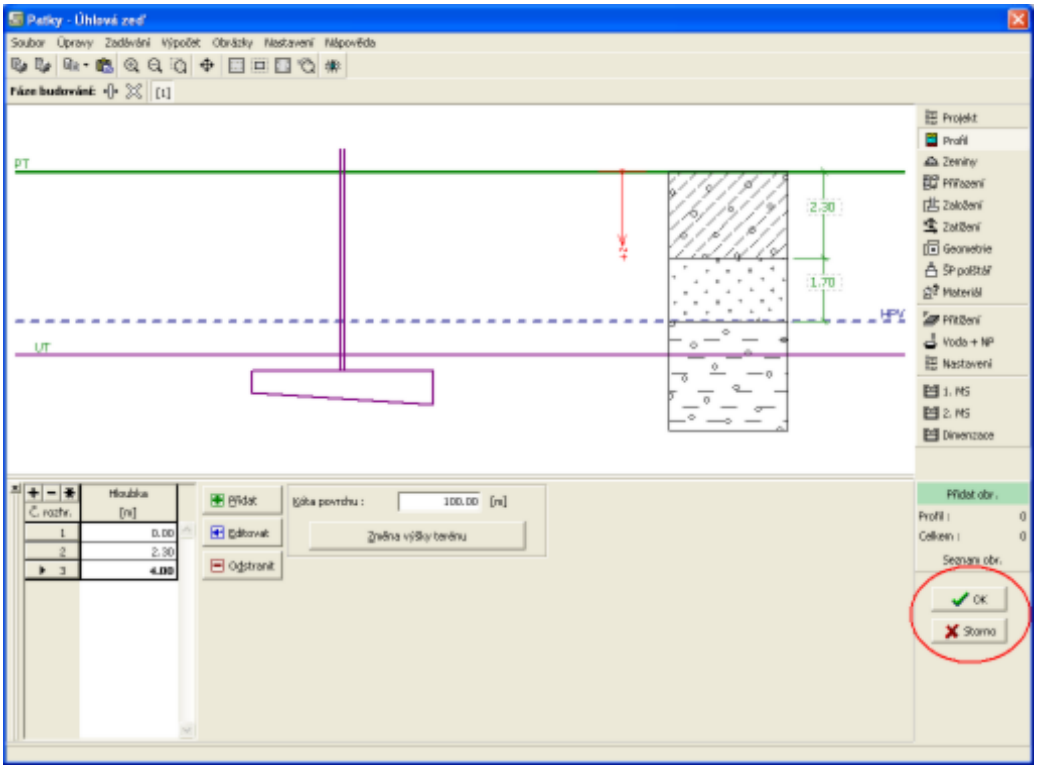
In some cases it is possible to launch a new program from a currently running program. For example, the "Cantilever wall" program allows for running the "Slope stability" program to verify the external stability of a structure, or the "Spread footing" program to verify the bearing capacity of a footing of a structure.

The new program loads the data of structure and then it behaves as a stand alone program – closing the program, however, is different. Pressing the "OK" button (on the right below the tool bars) closes the program and the analysis data are passed to the original program. This is not the case if closing the program with the help of the "Cancel" button.

The program, when running it for the first time, creates the data of a structure and passes on the structure dimensions, geology, loads, surcharges and other data. The program then requires inputting some additional data, e.g. the analysis method, analysis setting, slip surfaces, stages of construction, etc.

When running it again (always necessary if some changes were made in the original program) the program regenerates the data to be passed on, but keeps the data already inputted in this program. For example, when connecting the original program with the "Spread footing" program the new program keeps the additionally inputted sand-gravel cushion together with inputted soil – the footing dimensions, foundation geometry, and geological profile are, however, regenerated.

Some actions are not allowed in the new program – e.g. to change the basic setting of the project, unit, etc. The generated task, however, can be saved into new data using the "Save as" button and work with it as with any other independent task.



Run the program "Spread footing" from program "Cantilever wall"

Selecting and storing views

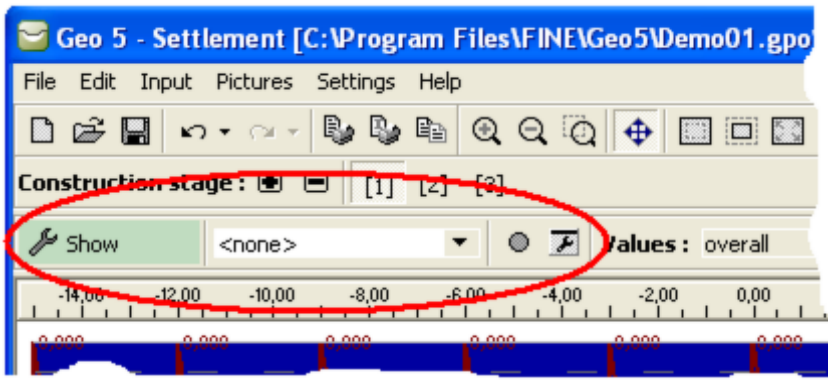
The programs offers a number of ways of displaying results. A specific option can be selected from the "View results" dialogue window. Quite often is necessary to go through a complex and tedious setting of views – for example, if we are interested in the distribution of internal forces developed in beams using FEM, it is necessary to turn off the color range, draw only undeformed structure, select a variable to be displayed, select a suitable magnification, etc.

To simplify the way of managing individual views the programs allow us, using the "Selecting and storing views" bar, to store the current view and also to go from one view to the other in a relatively simple way.

The stored view keeps:

- all settings from the "View results"
- drawn variables
- color range
- picture zoom

The view is stored for all stages of construction – if it is not possible in a certain construction stage to perform such a setting (e.g. in the first construction stage the settlement and depression are not defined) the programs displays the closest possible setting and the defined view is switched to <none>.


Geo 5 - Settlement [C:\Program Files\FINE\Geo5\Demo01.gpo
File Edit Input Pictures Settings Help
Construction stage :
[1] [2] [3]
Show
<none>
Values : overall
-14,00 -12,00 -10,00 -8,00 -6,00 -4,00 -2,00 0,00

Control units of a tool bar "Views"

The following control units are available to manage views:

Show	Define view	• opens the "Setting results visualization" dialogue window, which serves to define details for displaying the results
Stress	Select view	• a combo list allows for selecting an already specified and stored view
	Store current view	• opens the "New view" dialogue window to store a new view
	Open view manager	• opens the window with a list of views

Setting results visualization

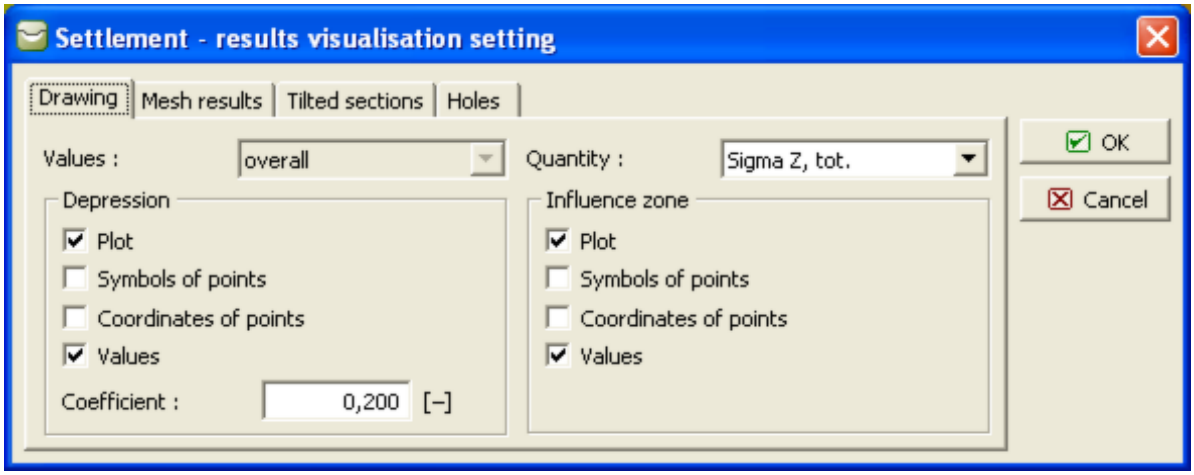
The "Settlement - results visualization setting " dialogue window provides tools for a lucid way of displaying the results both on the screen and in the printed format:

- parameters to draw depression line and influence zone
- setting surface views and color scale drawing
- setting and drawing tilted sections

The programs based on the finite element method further allow for setting:

- parameters to draw the finite element mesh
- parameters to draw construction – deformed / undeformed (note that undeformed option must be selected when displaying beam internal forces)
- distribution of internal forces along interfaces and on beam elements

All information specified in this window (including the setting of current magnification) can be stored using the selecting and storing views bar.

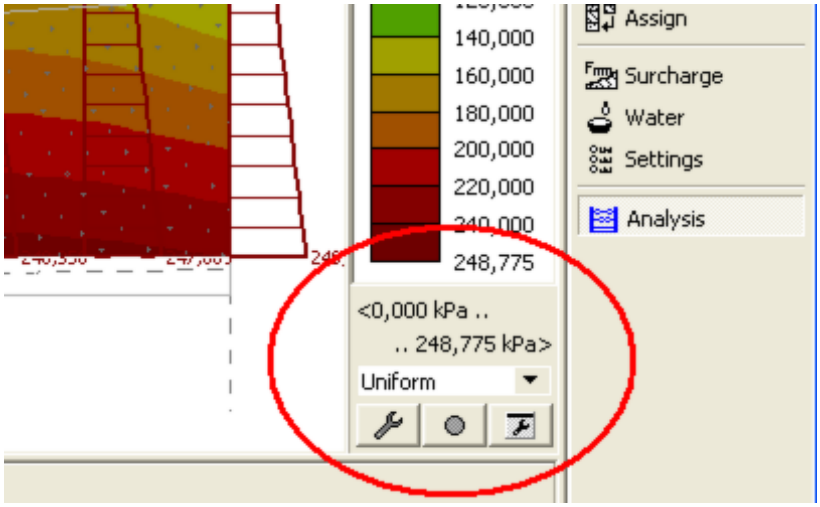


Dialogue window "Settlement – results visualization setting"

Setting color range

The color range is an important tool providing a lucid way of visualization of results. The program offers two predefined types of color ranges – " Uniform " and " Across zero ". Both ranges have a moving minimum and maximum value and predefined colors. The minimum and maximum values are automatically regenerated whenever the variable or a stage of construction is changed. The " Uniform " range means that colors are uniformly spread from the minimum to the maximum value. The " Across zero " range draws the positive values using warm colors (yellow, red), while cold colors (green, blue) are used to represent negative values.

The program allows for introduction of user-defined ranges with both the fixed minimum and maximum and the moving minimum and maximum. A user-defined range is specified in the "Scale color definition" dialogue window. The range is always defined for the current unit (e.g. *kPa, m*) – when switching the units the program always adjusts the range particular for a given unit.



Control units of a tool bar "Ranges"

The following control units are available to manage ranges:

Uniform	Select range	• a combo list allows for selecting an already specified and stored range
	Define color ranges	• opens the "Scale color definition" dialogue window to create a user-defined range

	Store current range	• opens the "New range" dialogue window to store a new range
	Open range manager	• opens the window with a list of automatic and user-defined ranges

Scale color definition

The "Scale color definition " dialogue window serves to cerate a user-defined color scale .

The "Floating minimum and maximum " check box determines the basic type of a scale. If checked, the minimum and maximum values of a scale are automatically adjusted whenever the corresponding variable or a construction stage is changed. In such a case it possible to adjust the following:

- scale refinement (the minimum number of levels is four and the maximum is 100)
- scale color
- uniform scale / across zero

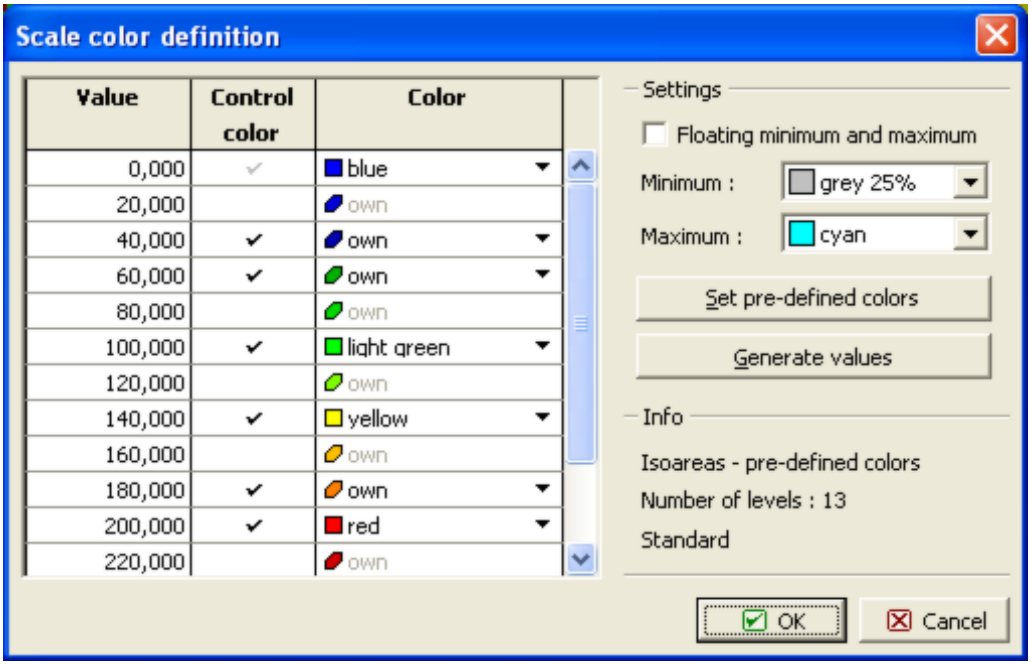
The number of scale levels and scale type are specified in the "Scale generation " dialogue window, which opens after pressing the " Generate values" button. It is possible to adjust both values and colors in the table in the left part of the dialogue window. The range values can be easily changed in the table. If the box in the " Control color " column is checked, it is possible to choose an arbitrary color from the combo box. The color on intermediate not checked rows are automatically blended from the inputted colors in checked rows. The default values can be recalled anytime after pressing the " Predefined colors " button.

An important property of the program is a definition of ranges with the fixed minimum and maximum. If the "Floating minimum and maximum " check box is not checked, the color range is fixed and its minimum and maximum values are inputted. As oppose to the moving range it is further possible to specify:

- range end points (in the "Scale generation" dialogue window)
- colors to display values out of the range

When changing a variable or a construction stage the color range remains still the same , keeping the same end ranges. The values found outside the range (below the minimum or above the maximum value) are drawn using colors specified in the right part of the window. The minimum and maximum range values are inputted in the "Scale generation " dialogue window. The inputted minimum and maximum values are linked to the same unit – e.g. when specifying the rage of *0-200 kPa* , this range is kept the same for all variables being specified in *kPa* – when changing the currently displayed variable to the variable settlement, the current range is switched to that corresponding to the unit of settlement.

For both the fixed and floating scale it is possible to choose whether the colors in the ranges are distributed uniformly or across zero. The "Uniform " scale means that colors are smoothly spread from the minimum to the maximum value of the scale. The " Across zero" scale draws the colors above the selected value using warm colors (yellow, red), while cold colors (green, blue) are used to represent the colors below the selected value.



Dialogue window "Scale color definition"

Input regimes and analysis

This capture contains basic description of individual regimes of inputting data into the program:

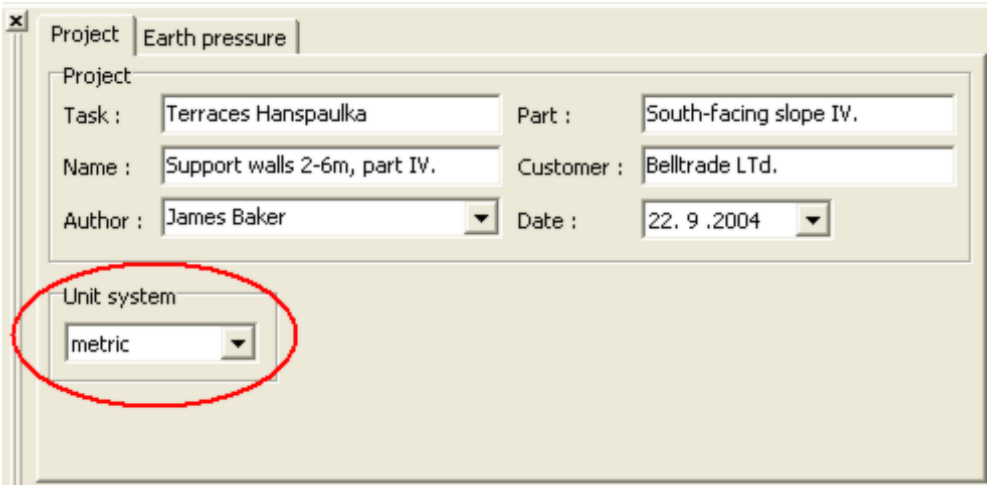
Program Earth Pressure

Project

The "Project" frame is used to input the basic project data and to specify the overall setting of the analysis run. The "Project" tab sheet contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc.

The "Project" tab sheet also allows the user to switch analysis units (metric / imperial).

The "Earth pressures" tab sheet serves to choose the basic theory or standards to be followed in the solution of a given problem.



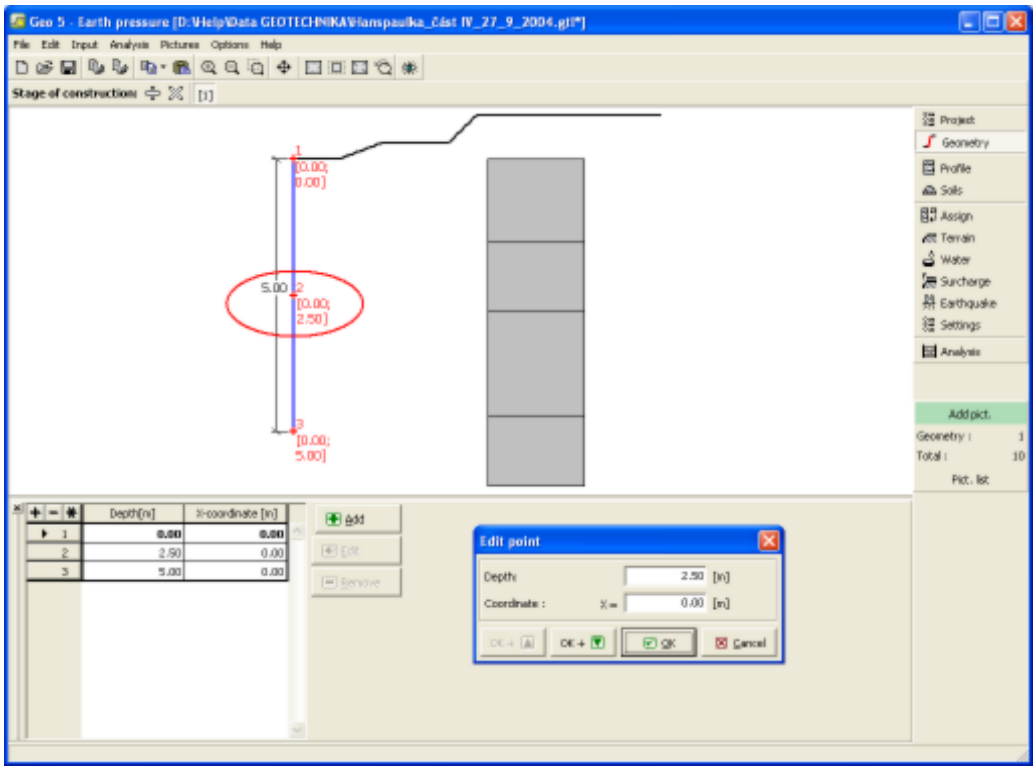
Frame "Project" - tab sheet "Project"

Geometry

The "Geometry" frame contains table listing the points of a structure. Adding (editing) points

is performed in the "Add (edit) point" dialog window.

The existing geometry points can be further edited on the desktop with the help of active objects – double clicking on a selected point opens a dialog window to edit the point.



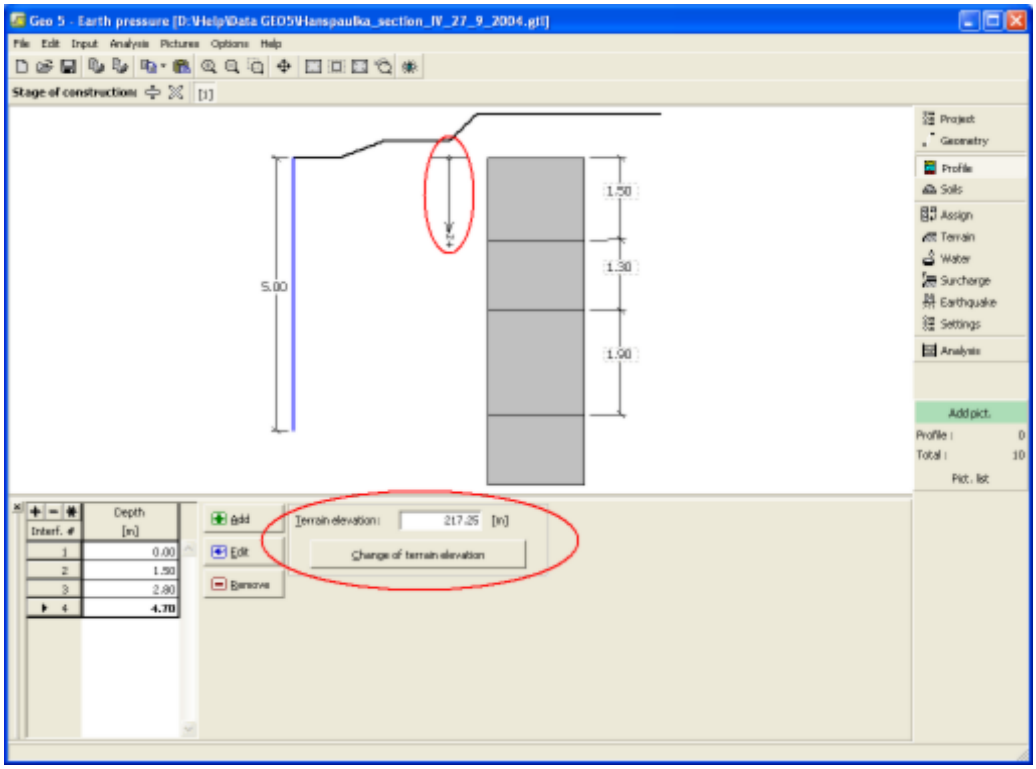
Frame "Geometry"

Profile

The "Profile" frame contains a table with a list of inputted interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "Add (edit) interface" dialogue window. The z-coordinate measured from the top point of a structure is specified.

The program allows for raising or lowering the top point of a structure in the "Change terrain elevation" dialogue window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".



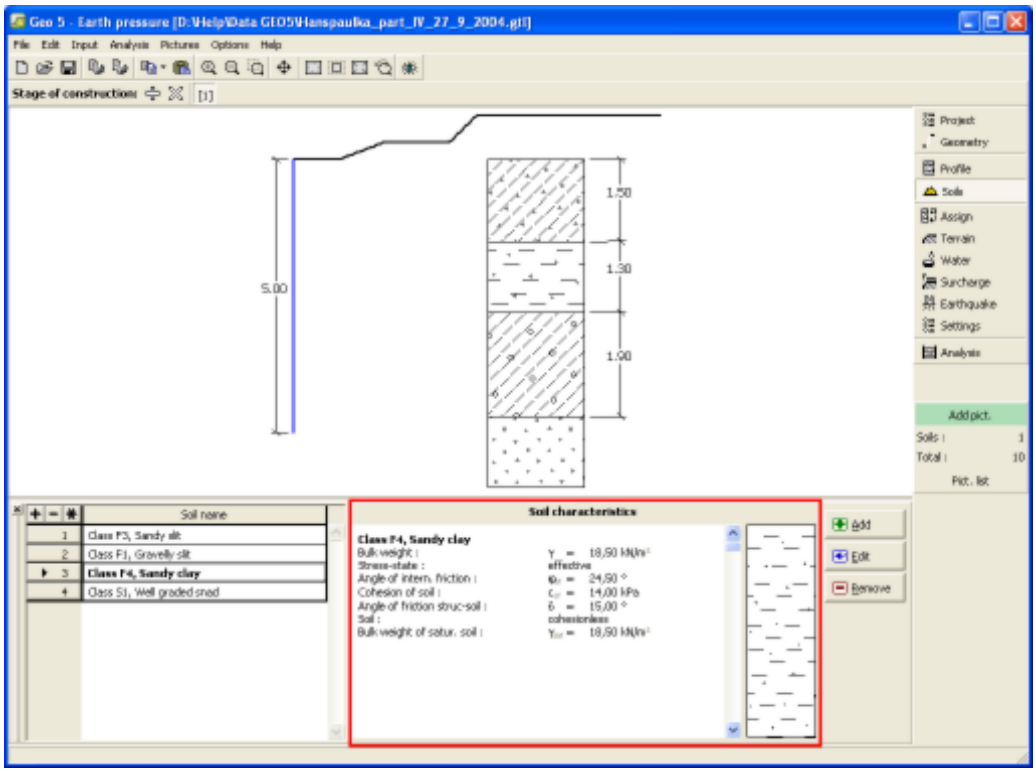
Frame "Profile"

Soils

The "Soils" frame contains a table with a list of inputted soils. The table also provides information about currently selected soil displayed in the right part of the frame.

Adding (editing) a soil is performed in the "Add (edit) soil" dialogue window.

The soil characteristics are specified in the program "Earth Pressure". These characteristics are further specified in chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

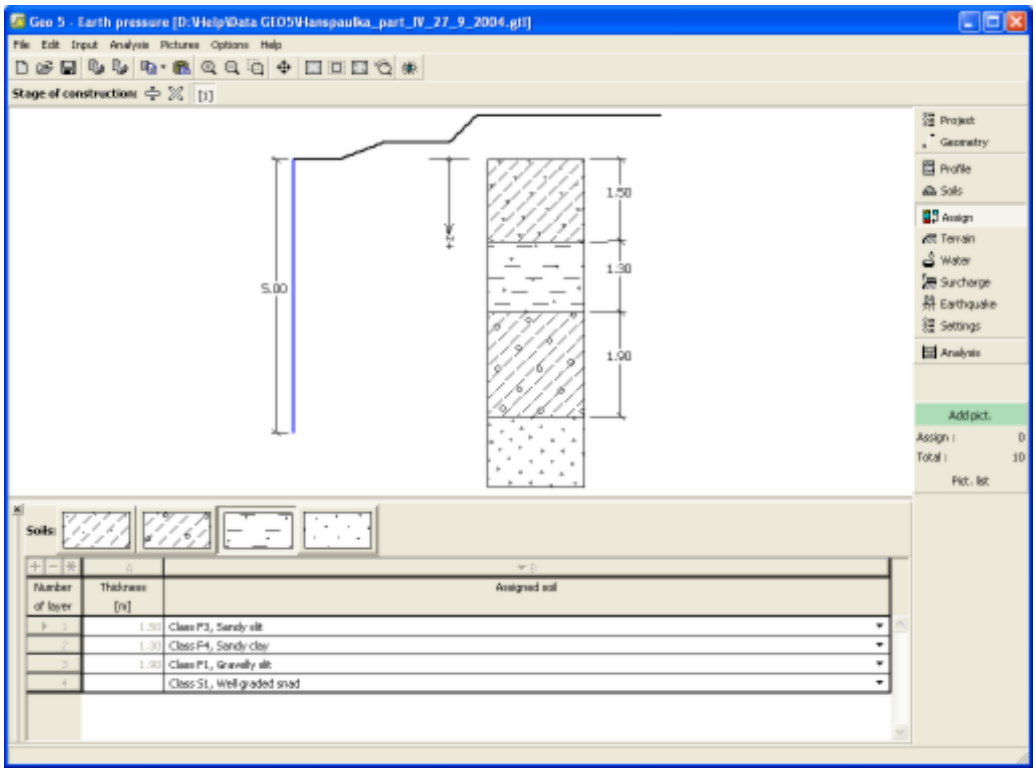


Frame "Soils"

Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

Procedure to assign soil into a layer is described in details herein.



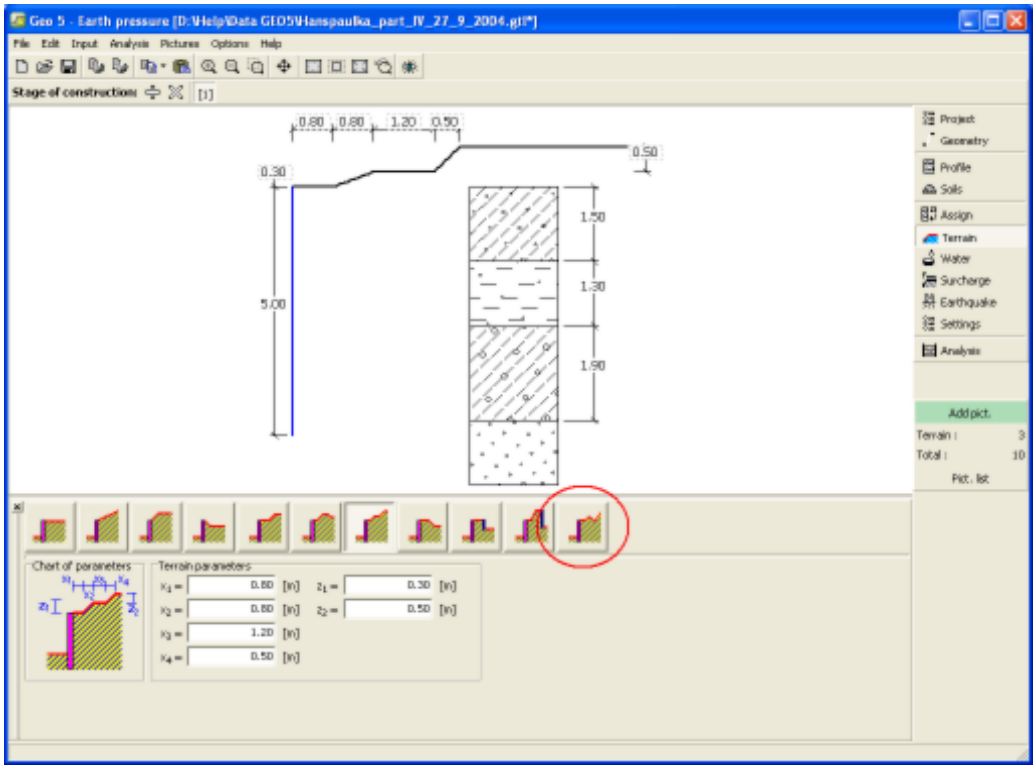
Frame "Assign"

Terrain

The "Terrain" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0, 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

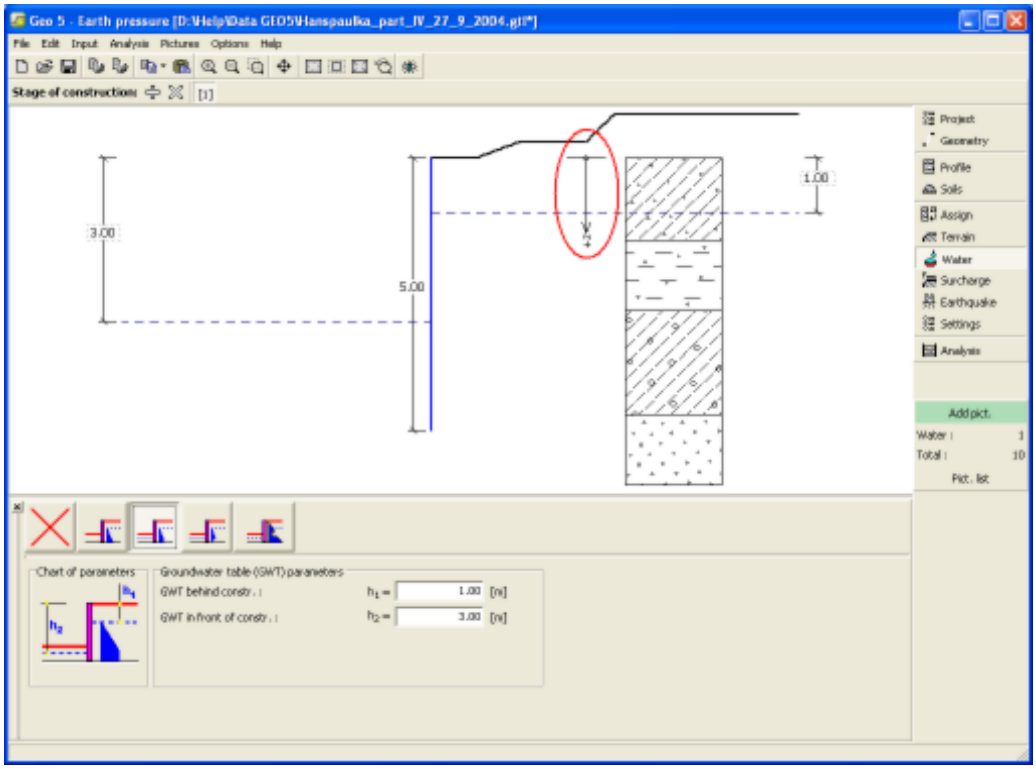
The "Water" allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint (“Parameter chart”) of inputted values is displayed in the left part of the frame. Water parameters (h1, h2...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option is a manual input of pore pressure both in front and behind the structure. Two tab sheets "In front of structure" and "Behind structure" appear with tables. The table is filled with values of pore pressure in front, or behind the structure at a depth of “z” (z-axis).

The ground water table can also be specified above the structure or earth profile, respectively – in such a case the depth of water is inputted with negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the hint chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



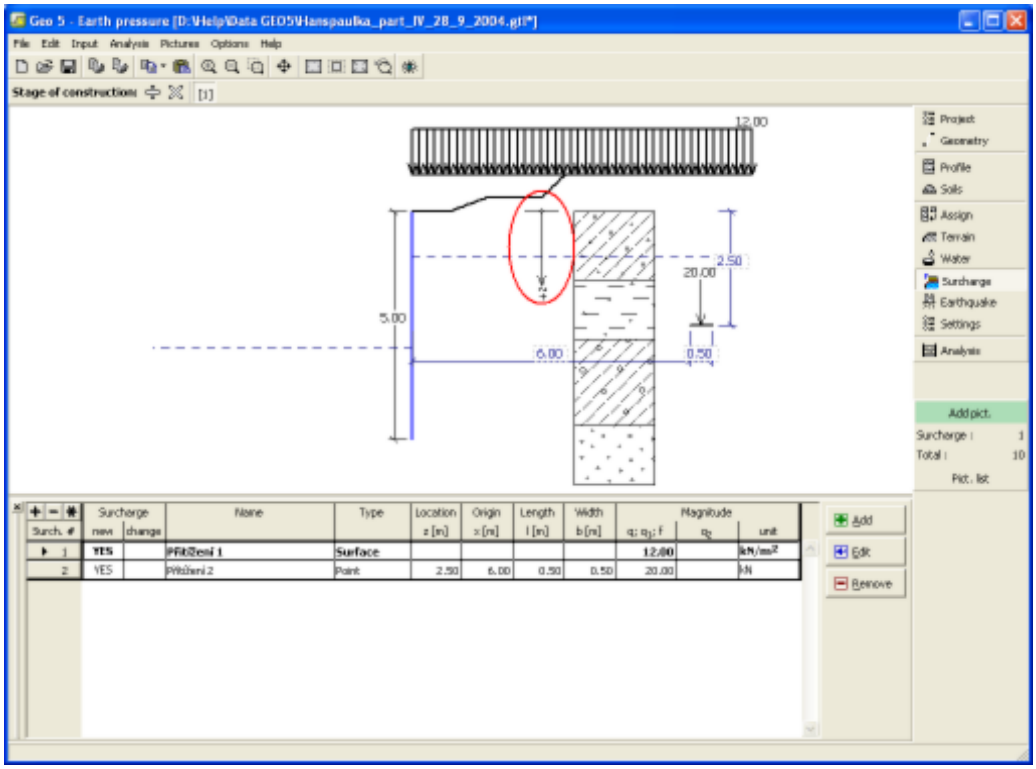
Frame "Water"

Surcharge

The "Surcharge" frame contains a table with a list of inputted surcharges. Adding (editing) surcharge is performed in the "New (edit) surcharge" dialogue window. The inputted surchages can be edited on the desktop with the help of active dimensions or active objects, respectively.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. Providing the surcharge is found off the terrain the computer prompts an error message.

Analysis of earth pressures due to surcharges is described in the theoretical part of the hint, chapter "Influence of surcharge".

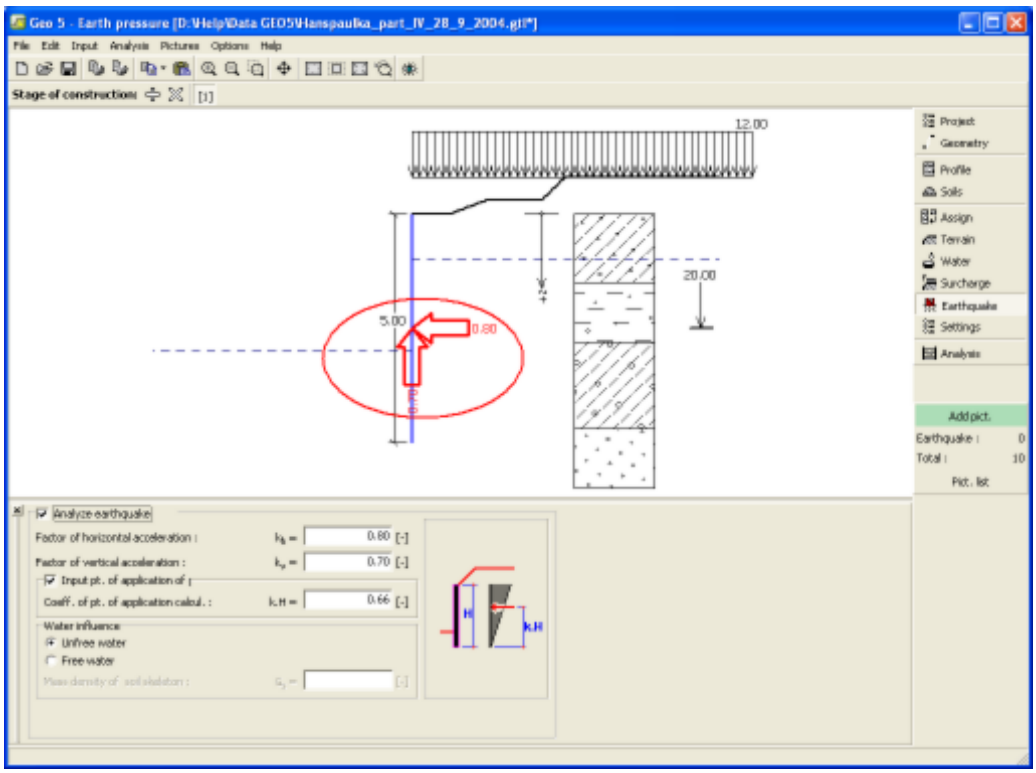


Frame "Surcharge"

Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of inputted earthquake effects are displayed on the desktop.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the hint in chapter "Influence earthquake".

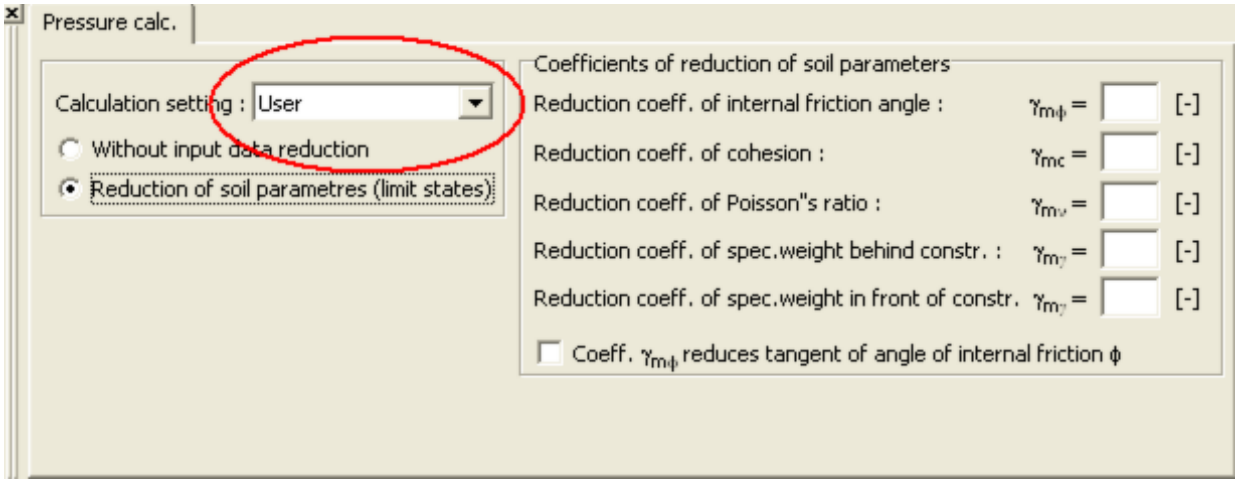


Frame "Earthquake"

Setting

The "Setting" frame contains basic settings for the analysis of earth pressures. The program offers pre-setting for different countries (Czech Republic, Germany, France,...) and "Standard setting" recommended by the authors of the program for countries no included in the list. While changing settings in the combo list the values of coefficients of reduction of soil parameters in corresponding windows are changed.

An arbitrary analysis setting is available with the option "User setting". Selecting the option "Reduce soil parameters" allows for specifying in input fields individual values of the coefficients of reduction of soil parameters (e.g., recommended values according to EC7).


Pressure calc.
Calculation setting : User
Without input data reduction
Reduction of soil parametres (limit states)
Coefficients of reduction of soil parameters
Reduction coeff. of internal friction angle : γmφ = [-]
Reduction coeff. of cohesion : γmc = [-]
Reduction coeff. of Poisson"s ratio : γmν = [-]
Reduction coeff. of spec.weight behind constr. : γmγ = [-]
Reduction coeff. of spec.weight in front of constr. γmγ = [-]
Coeff. γmφ reduces tangent of angle of internal friction φ

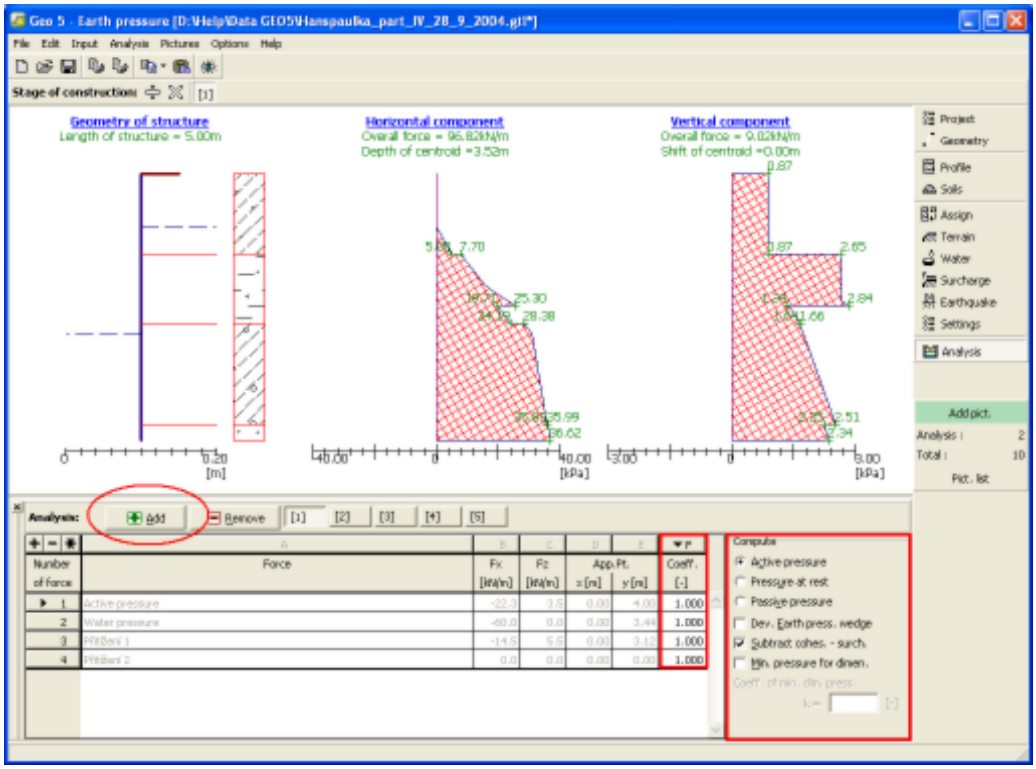
Frame "Setting"

Analysis

The "Analysis" frame displays the analysis results. The frame serves to select type of computed earth pressure (active pressure, pressure at rest, passive pressure). Two options "Create soil wedge" and "Minimum dimensioning pressure" are available when computing the active earth pressure.

Several analyses can be performed for a single task by varying design coefficients of individual components of earth pressure.

The analysis results are displayed on the desktop and are updated im mediately for an arbitrary change in input data or setting. Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Analysis"

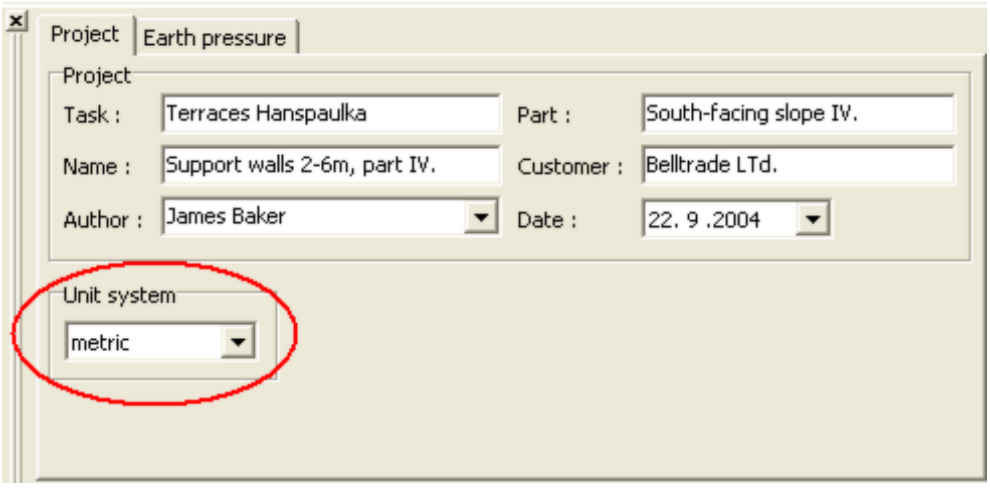
Program Sheeting Design

Project

The "Project" frame is used to input the basic project data and to specify the overall setting of the analysis run. The "Project" tab sheet contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc.

The "Project" tab sheet also allows the user to switch analysis units (metric / imperial).

The "Earth pressures" tab sheet serves to choose the basic theory or standards to be followed in the solution of a given problem.



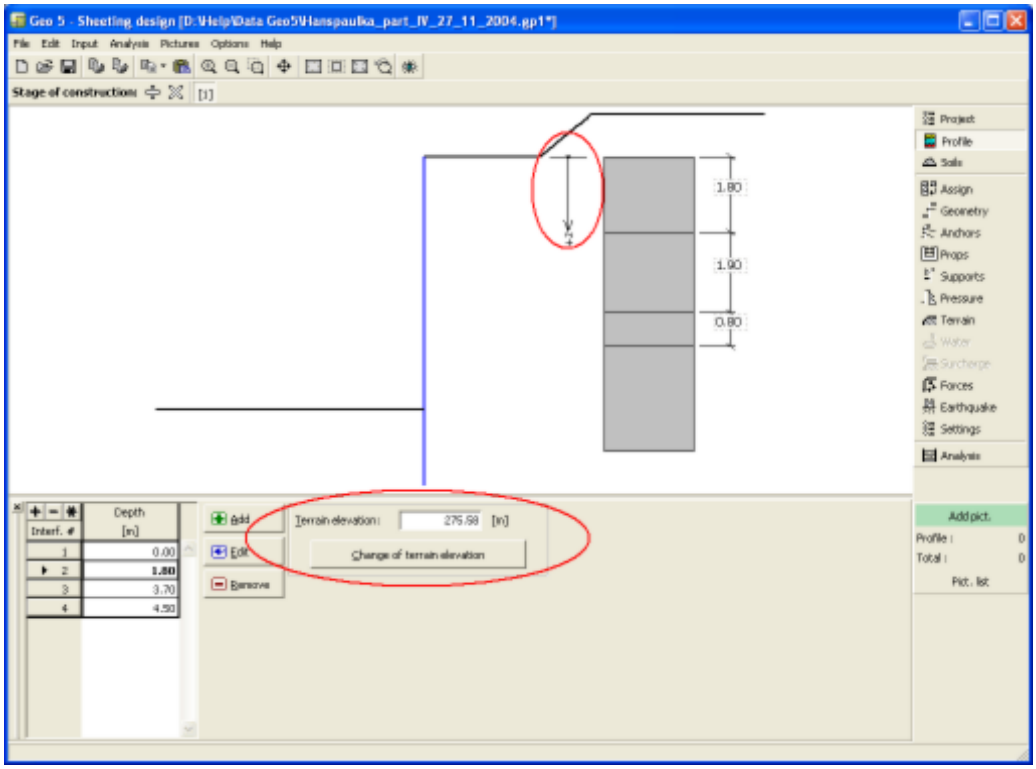
Frame "Project" - tab sheet "Project"

Profile

The "Profile" frame contains a table with a list of inputted interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "Add (edit) interface" dialogue window. The z-coordinate measured from the top point of a structure is specified.

The program allows for raising or lowering the top point of a structure in the "Change terrain elevation" dialogue window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".



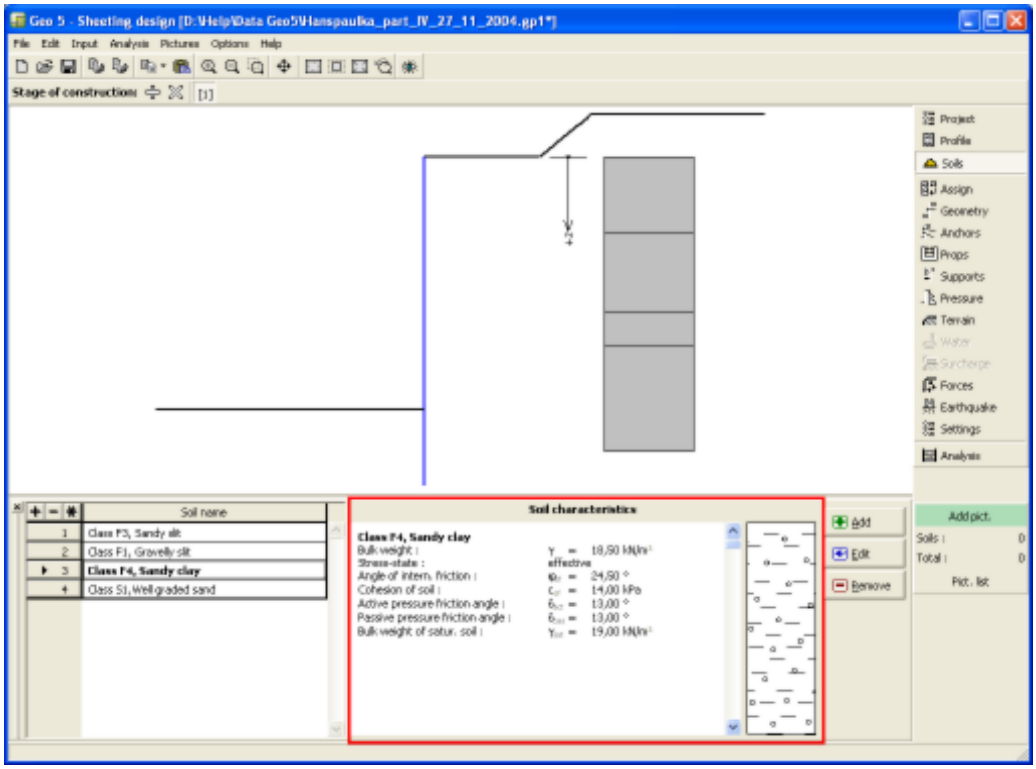
Frame "Profile"

Soils

The "Soils" frame contains a table with a list of inputted soils. The table also provides information about currently selected soil displayed in the right part of the frame.

Adding (editing) a soil is performed in the "Add (edit) soil" dialogue window.

The soil characteristics are specified in the program "Sheeting design". These characteristics are further specified in chapters: "Basic data" and "Uplift pressure".

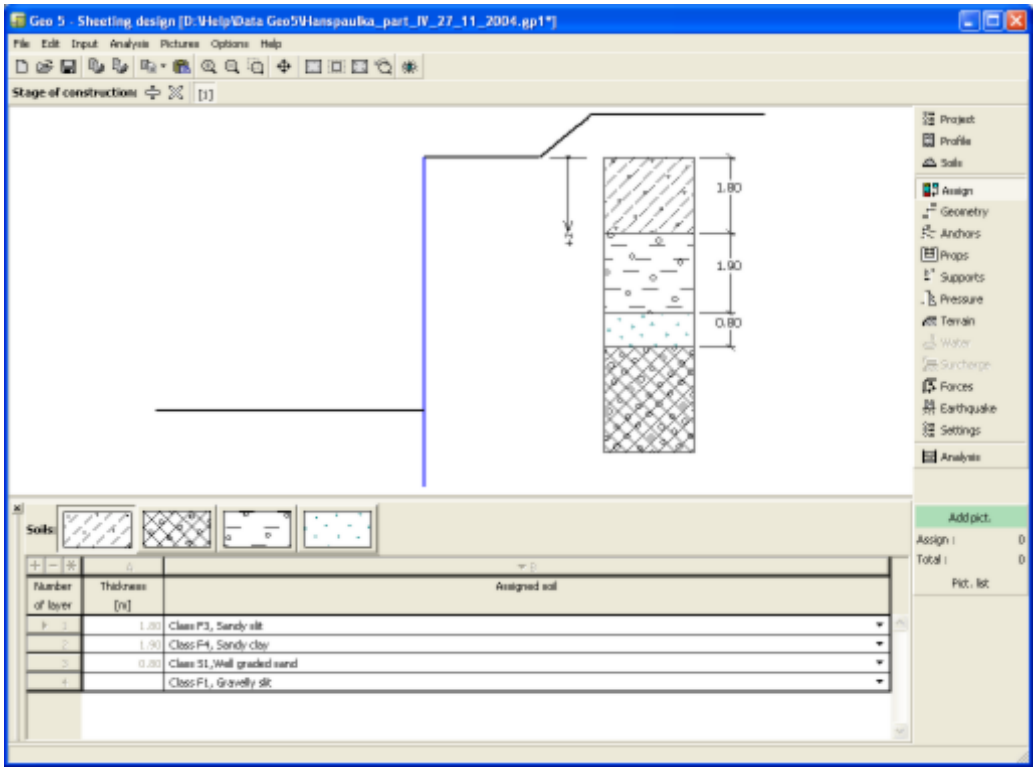


Frame "Soils"

Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

Procedure to assign soil into a layer is described in details herein.

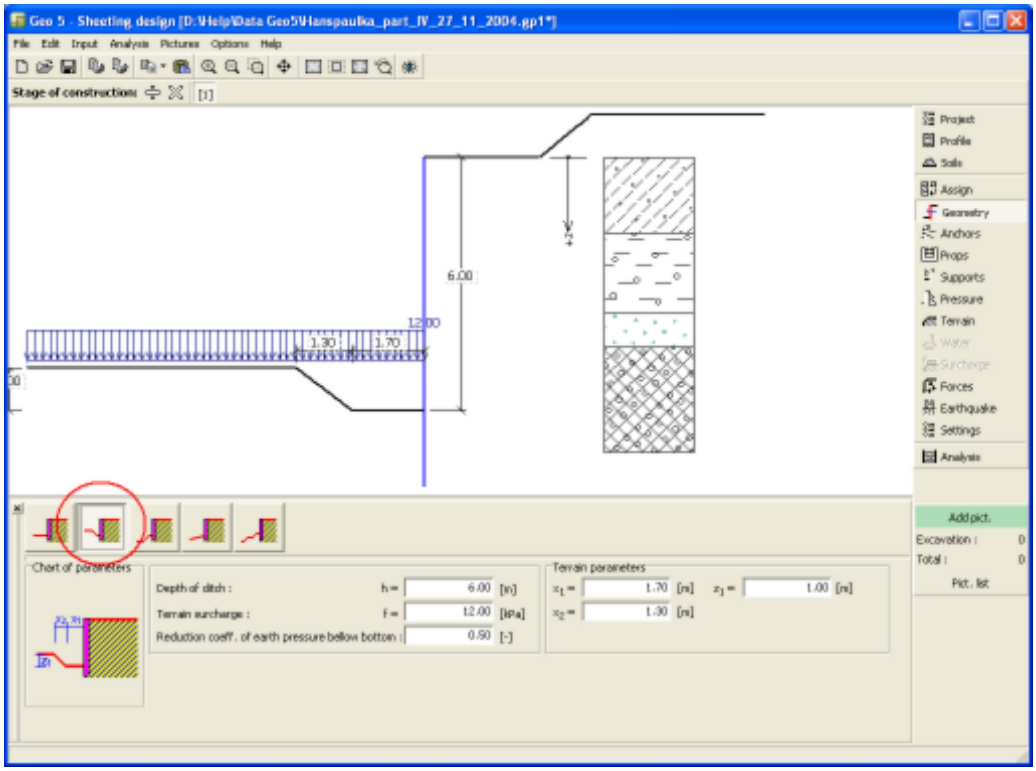


Frame "Assign"

Geometry

The "Geometry" frame is used to specify the depth of a construction ditch and by pressing the button to choose the shape of a bottom. The selected shape with a graphic hint ("Parameter chart") appears in the left part of the frame. The dimensions of a structure can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

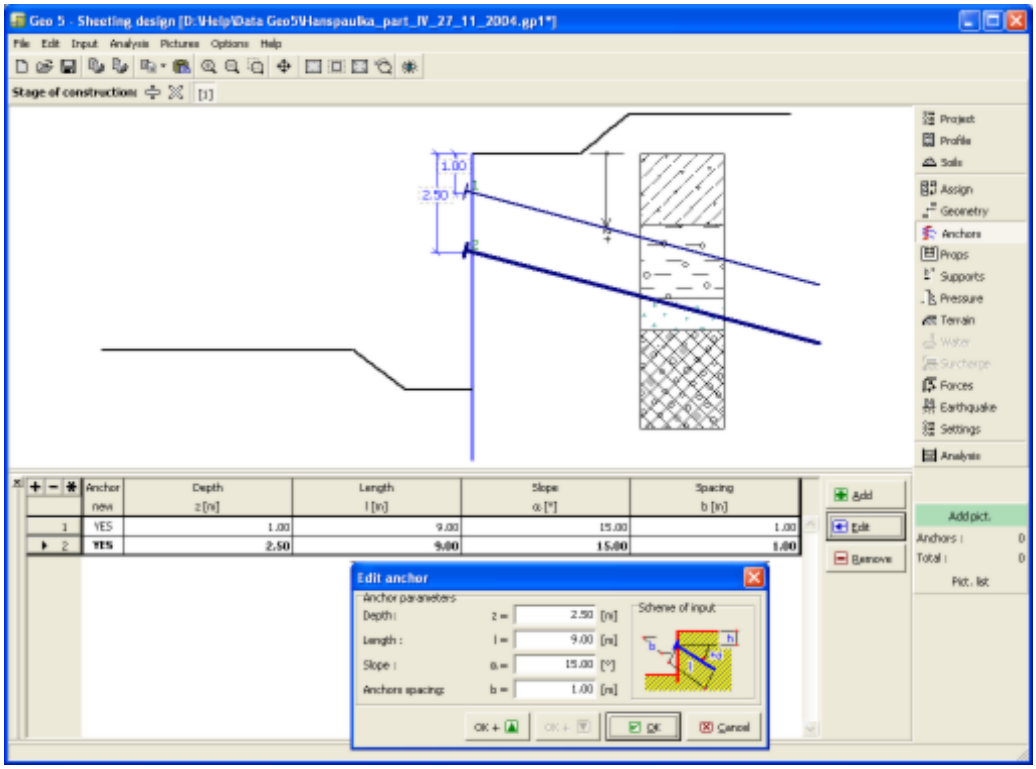
The frame can be further used to input surcharge of a construction ditch bottom and coefficient of reduction of earth pressure below the ditch bottom (this coefficient serves to analyze braced sheeting).



Frame "Geometry"

Anchors

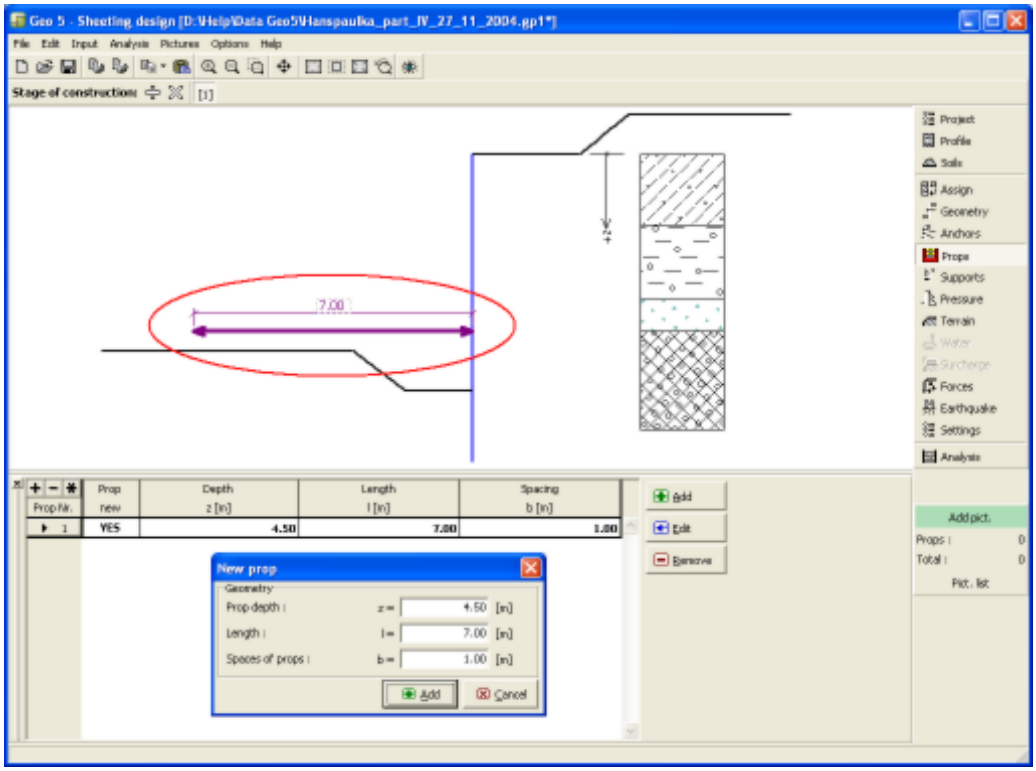
The "Anchors" frame contains a table with a list of inputted anchors. Adding (editng) anchors is performed in the "New anchor (Edit anchor)" dialogue window. The inputted anchors can be edited on the desktop with the help of active objects.



Frame "Anchors"

Props

The "Props" frame contains a table with a list inputted props. Adding (editing) props is performed in the "New prop (Edit prop)" dialogue window. The inputted props can also be edited on the desktop with the help of active dimensions or active objects, respectively.

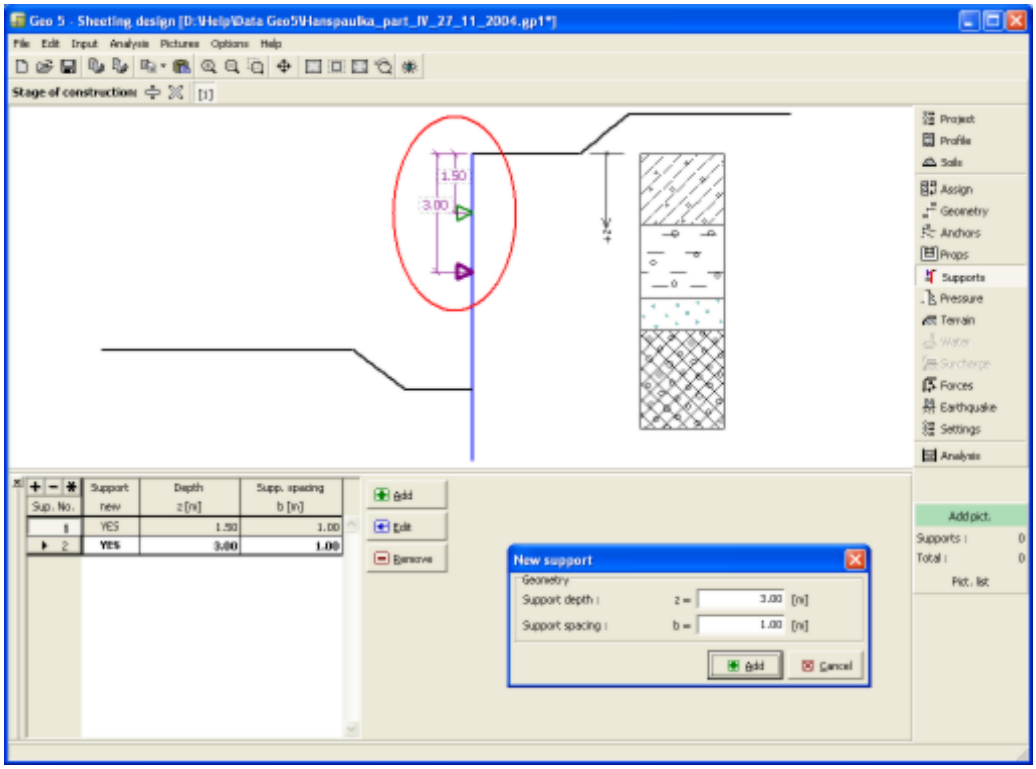


Frame "Props"

Supports

The "Supports" frame contains a table with a list of inputted supports. Adding (editing)

supports is performed in the "New support (Edit support)" dialogue window. The inputted supports can also be edited on the desktop with the help of active dimensions or active objects, respectively.

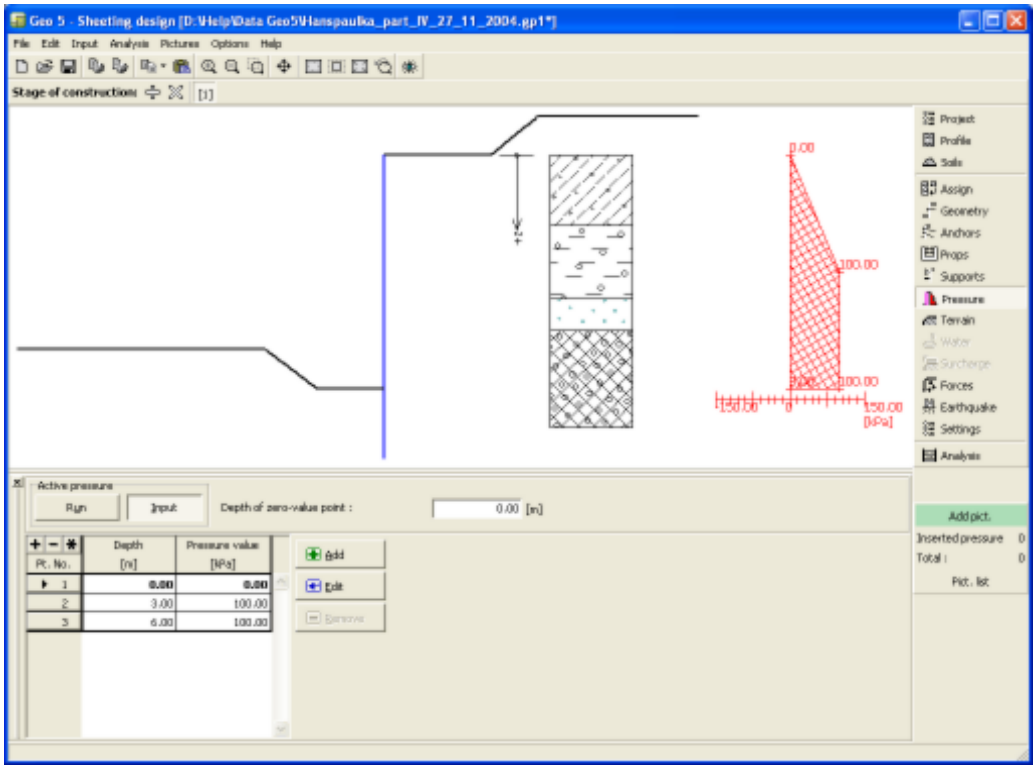


Frame "Supports"

Pressure determination

The "Pressure specification" frame allows by pressing the button "Analyze" ("Input", respectively) for selecting a method for the calculation of active earth pressure. Choose option "Analyze" if you wish the active earth pressure to be computed automatically based on specified earth profile.

In some special cases (redistribution of earth pressures due to presence of anchors, nonstandard rotation of a structure) it advisable to specify the distribution of earth pressure on a structure manually. Selecting the option "Input" opens a table in the frame with a list of inputted points and the corresponding pressure value. The pressure is specified up to the depth of structure increased by the depth of zero point (the depth of zero point is introduced in the top part of the frame). The depth of zero point equal to zero is selected if we wish to specify the pressure values only up to the depth of construction ditch. Below the ditch the programs computes the pressure values based on the specified geological profile. Providing the earth pressure is specified manually the program does not account for the influence of terrain profile, surcharge and water.



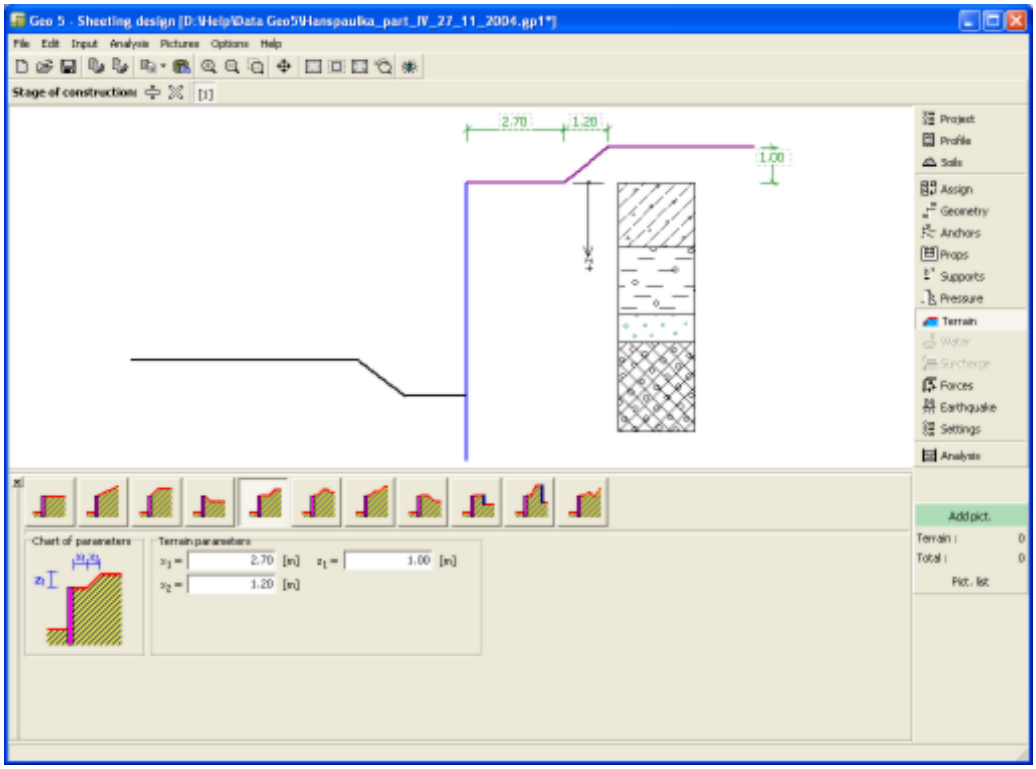
Frame "Pressure determination"

Terrain

The "Terrain" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0, 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the hint, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

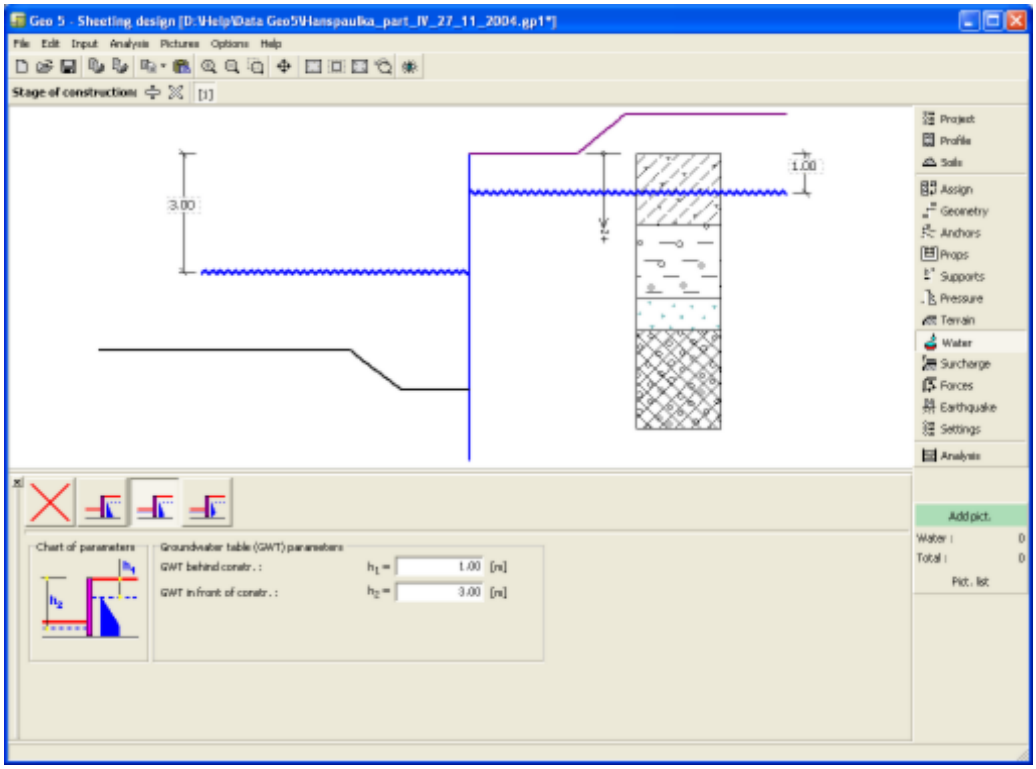
Water

The "Water" allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. Water parameters h_1, h_2 can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The ground water table can also be specified above the structure or earth profile, respectively – in such a case the depth of water is inputted with negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the hint chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



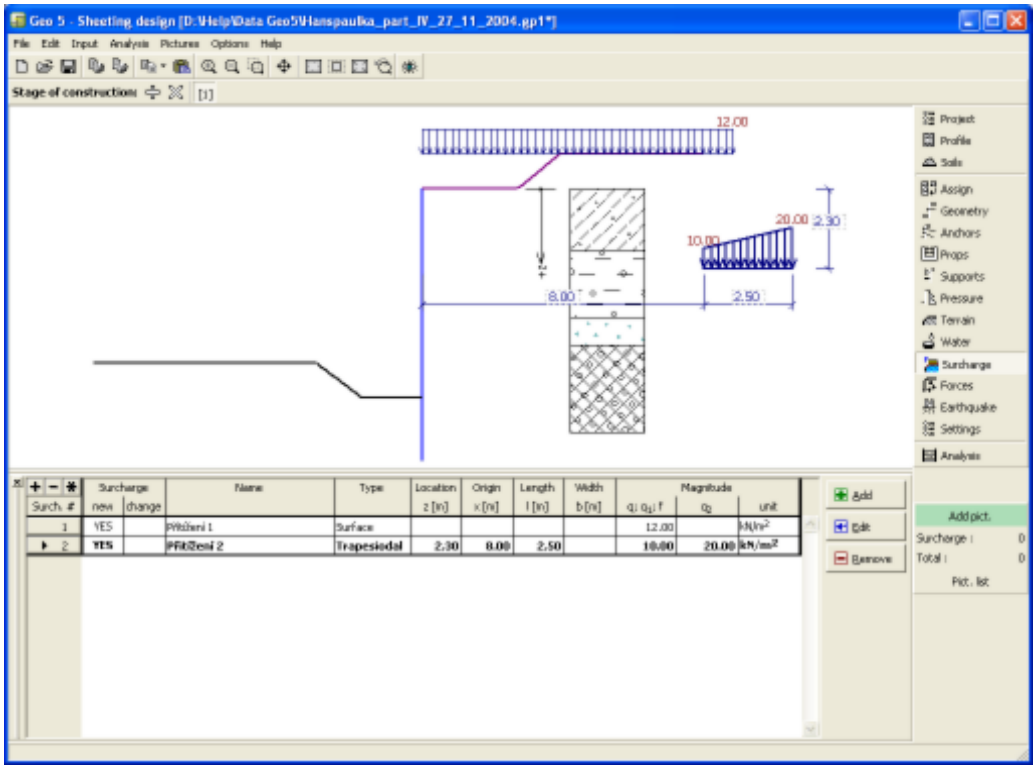
Frame "Water"

Surcharge

The "Surcharge" frame contains a table with a list of inputted surcharges. Adding (editing) surcharge is performed in the "New (edit) surcharge" dialogue window. The inputted surchages can be edited on the desktop with the help of active dimensions or active objects, respectively.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. Providing the surcharge is found off the terrain the computer prompts an error message.

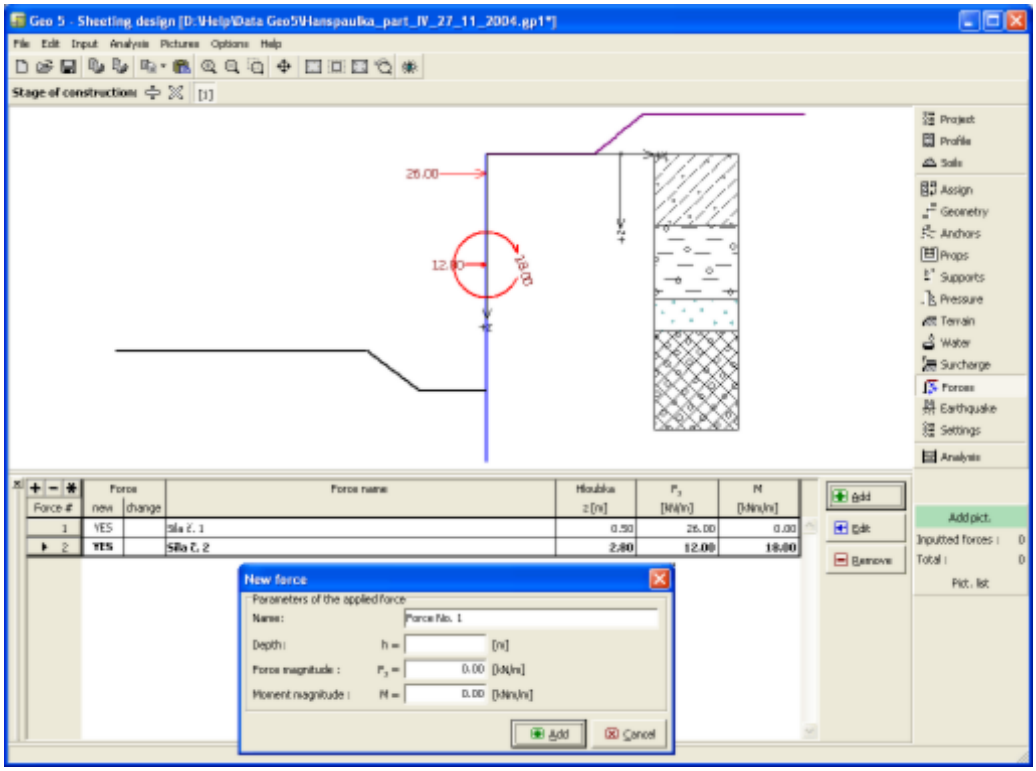
Analysis of earth pressures due to surcharges is described in the theoretical part of the hint, chapter "Influence of surcharge".



Frame "Surcharge"

Forces

The "Forces" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "New force (edit force)" dialogue window. The inputted forces can also be edited on the desktop with the help of active objects.



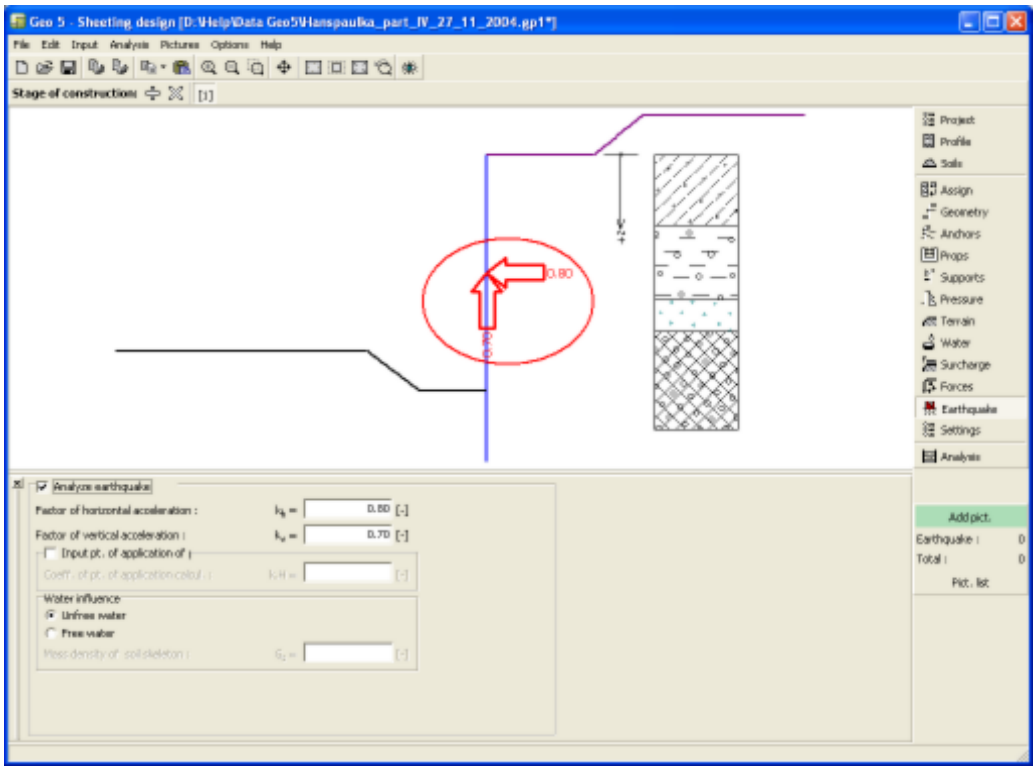
Frame "Forces"

Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of inputted

earthquake effects are displayed on the desktop.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the hint in chapter "Influence earthquake".

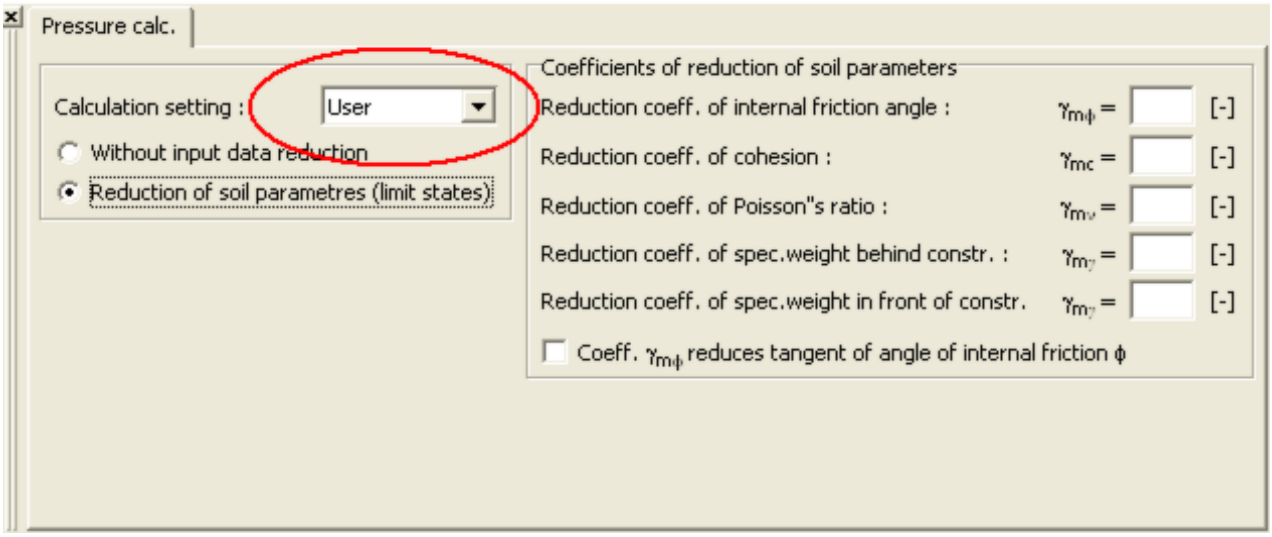


Frame "Earthquake"

Setting

The "Setting" frame contains basic settings for the analysis of earth pressures. The program offers pre-setting for different countries (Czech Republic, Germany, France,...) and "Standard setting" recommended by the authors of the program for countries no included in the list. While changing settings in the combo list the values of coefficients of reduction of soil parameters in corresponding windows are changed.

An arbitrary analysis setting is available with the option "User setting". Selecting the option "Reduce soil parameters" allows for specifying in input fields individual values of the coefficients of reduction of soil parameters (e.g., recommended values according to EC7-1).


Pressure calc.
Calculation setting :
User
Without input data reduction
Reduction of soil parametres (limit states)
Coefficients of reduction of soil parameters
Reduction coeff. of internal friction angle :
Reduction coeff. of cohesion :
Reduction coeff. of Poisson"s ratio :
Reduction coeff. of spec.weight behind constr. :
Reduction coeff. of spec.weight in front of constr.
Coeff. reduces tangent of angle of internal friction ϕ
[-]

Frame "Setting"

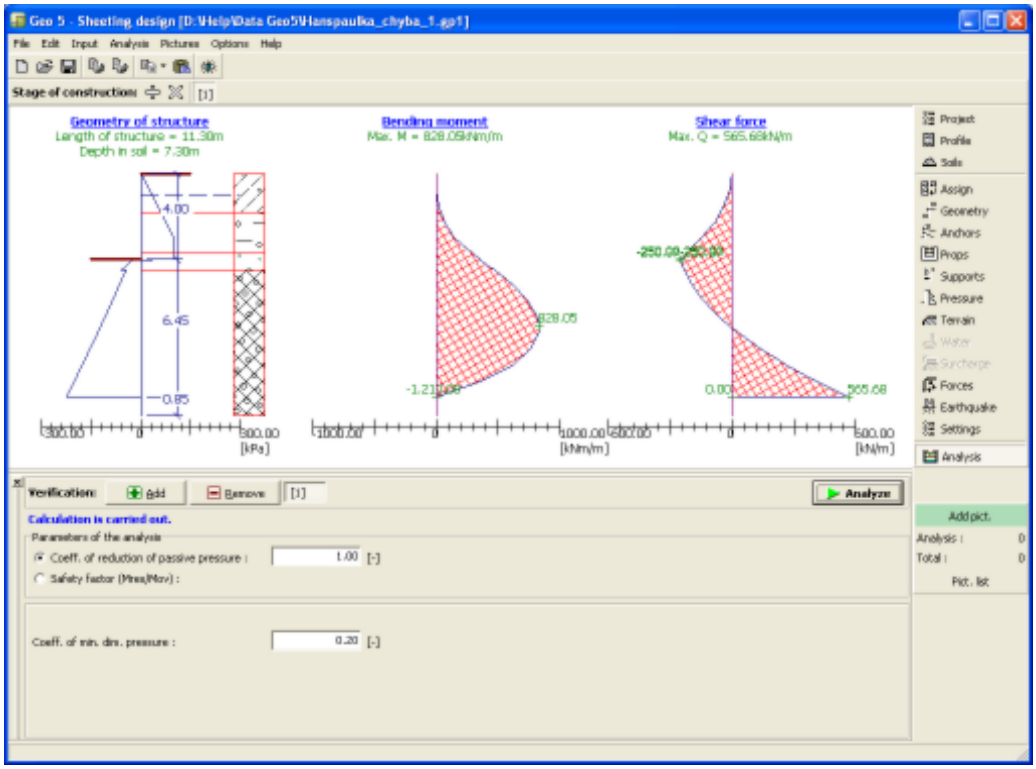
Analysis

The "Analysis" frame displays the analysis results. The analysis is carried out by pressing the "Analyze" button in the right part of the frame. The frame has two variants. The first variant applies to a wall free of anchors (sheet pile) and the second one to an anchored (strutted) wall.

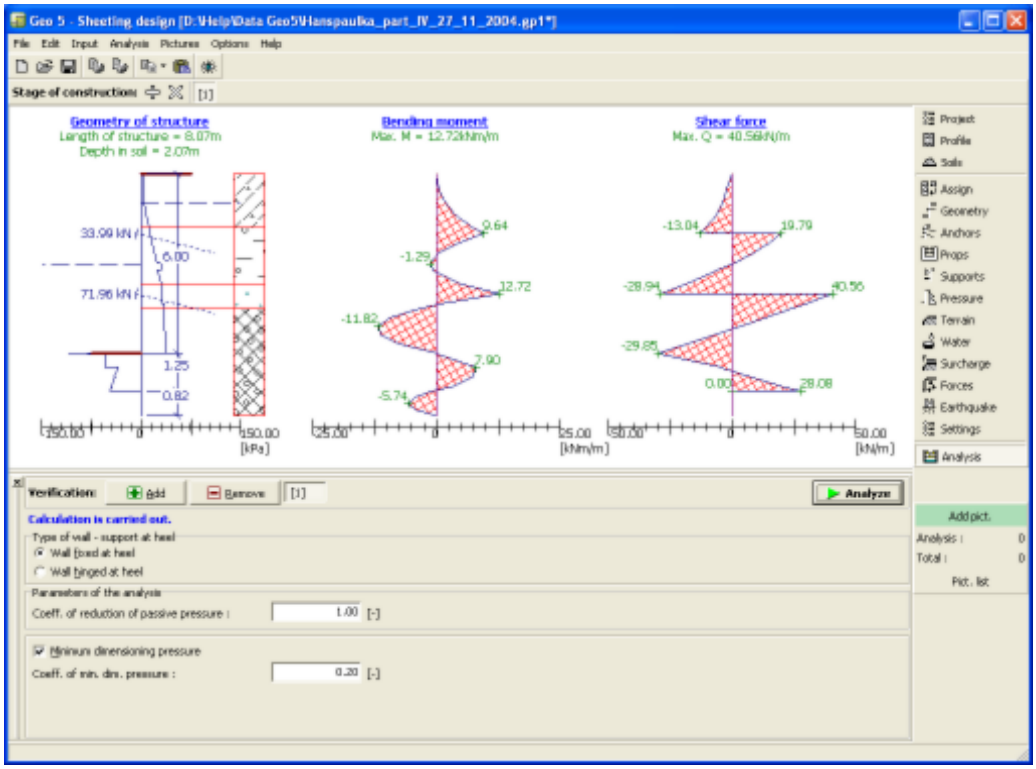
A coefficient of reduction of passive earth pressure (or factor of safety) together with a choice whether to consider a minimum dimensioning pressure behind the structure is specified for a non-anchored wall.

A type of heel support (fixed, free) and analysis parameters (coefficient of reduction of passive pressure, minimum dimensioning pressure) are specified for an anchored wall.

The analysis results are displayed on the desktop. Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Analysis" - non-anchored wall



Frame "Analysis" - anchored wall

Program Sheeting Check

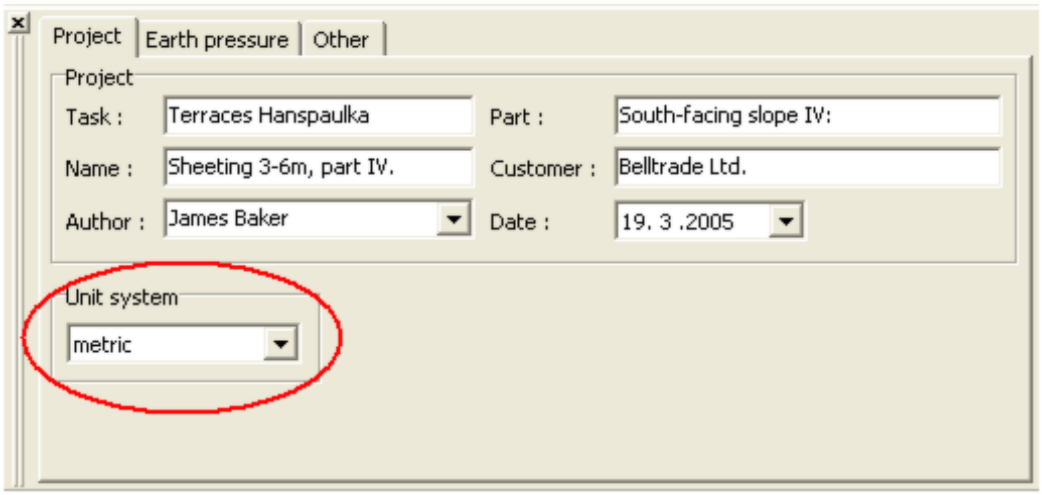
Project

The "Project" frame is used to input the basic project data and to specify the overall setting of the analysis run. The "Project" tab sheet contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc.

The "Project" tab sheet also allows the user to switch analysis units (metric / imperial).

The "Earth pressures" tab sheet serves to choose the basic theory or standards to be followed in the solution of a given problem.

The "Other" tab sheet allows the user to specify subdivision of a wall in to finite elements (by default the number of elements equals to 30).



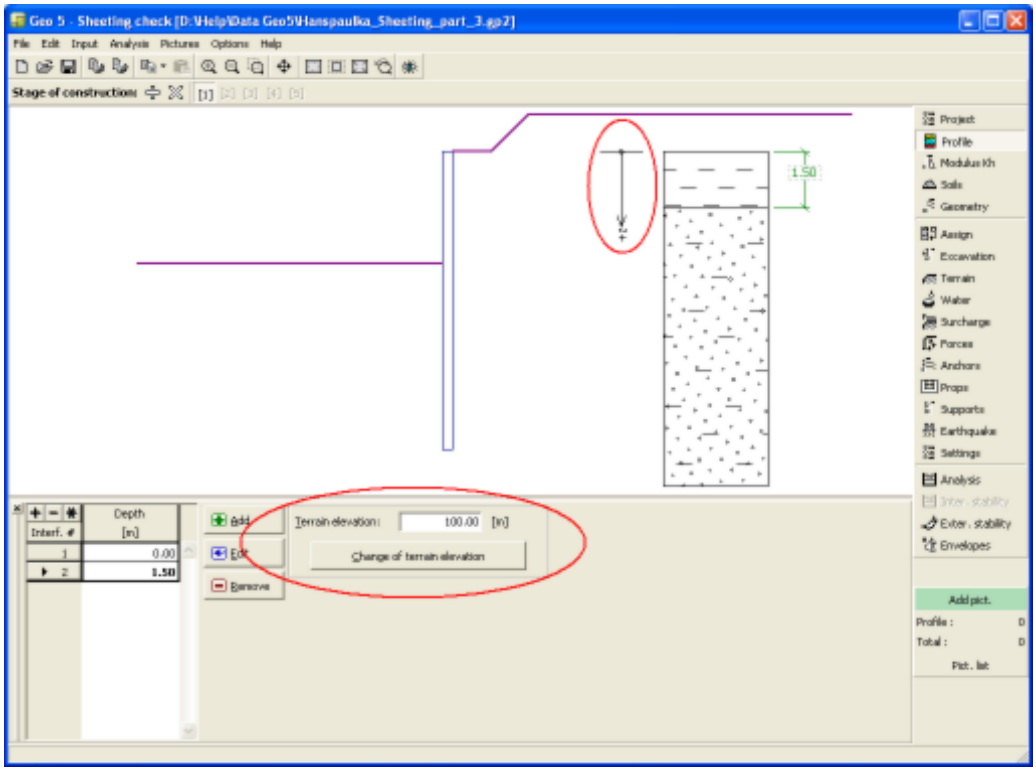
Frame "Project" - tab sheet "Project"

Profile

The "Profile" frame contains a table with a list of inputted interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions .

Adding (editing) layer is performed in the "Add (edit) interface" dialogue window. The z-coordinate measured from the top point of a structure is specified.

The program allows for raising or lowering the top point of a structure in the "Change terrain elevation" dialogue window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".



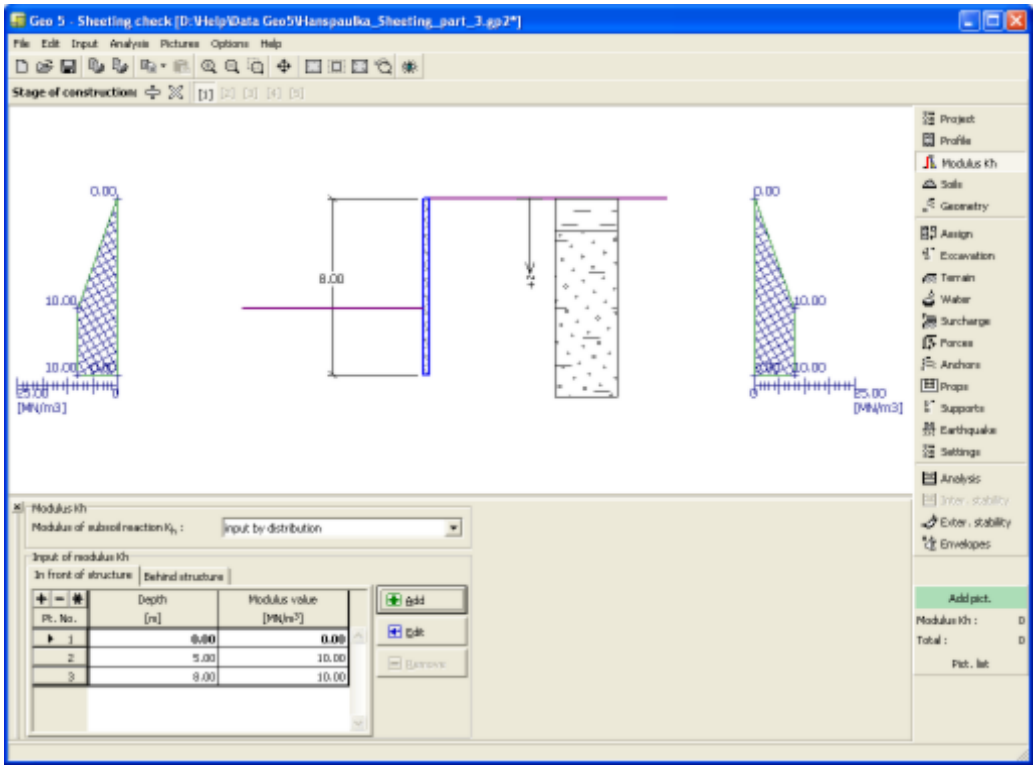
The frame "Profile"

Modulus of subsoil reaction

This frame serves to specify a type of analysis for computation of the modulus of subsoil reaction, which is an important input parameter when analyzing a structure using the method of dependent pressures.

The program makes possible to input the distribution of the modulus of subsoil reaction (along the length of a structure, as a soil parameter), iterate from soil material parameters, or to compute it. The modulus of subsoil reaction can be either linear or nonlinear.

Selecting the option "Input by distribution" opens a table in the frame that allows for specifying the values of the modulus of subsoil reaction both in front and behind the structure. When selecting other options the required information needed to compute the modulus value are inputted as soil parameters in the frame "Soils".



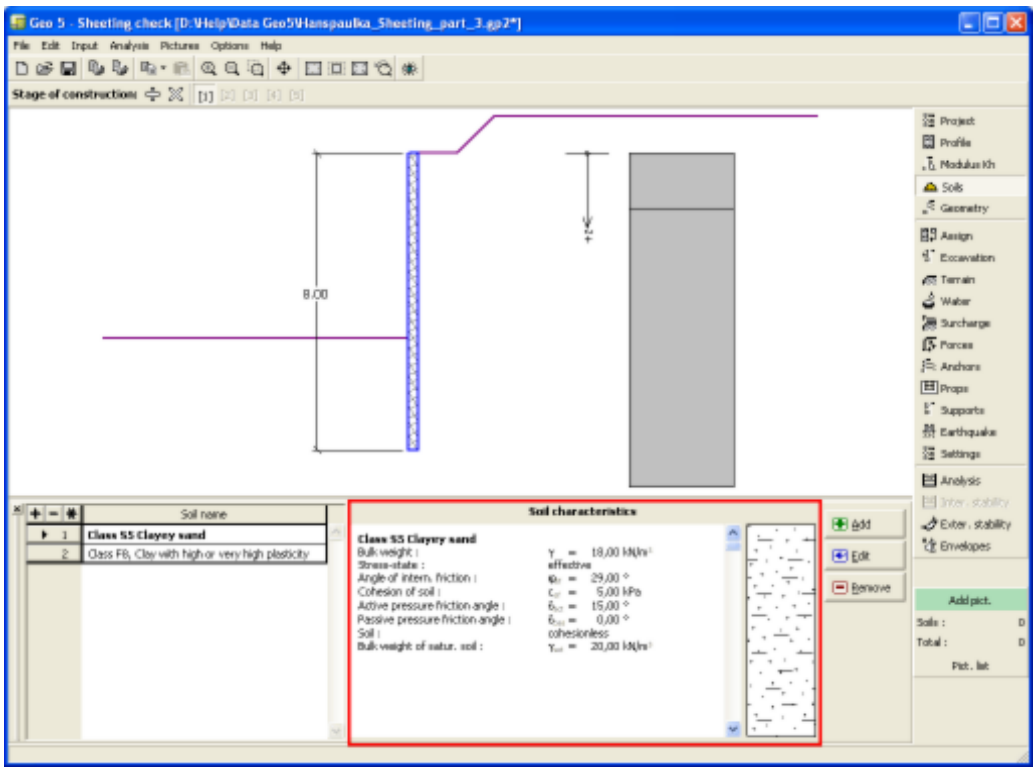
Frame "Modulus of subsoil reaction"

Soils

The "Soils" frame contains a table with a list of inputted soils. The table also provides information about currently selected soil displayed in the right part of the frame.

Adding (editing) a soil is performed in the "Add (edit) soil" dialogue window.

The soil characteristics are specified in the program "Sheeting check". These characteristics are further specified in chapters: "Basic data", "Earth pressure at rest", "Uplift pressure" and "Modulus of subsoil reaction".



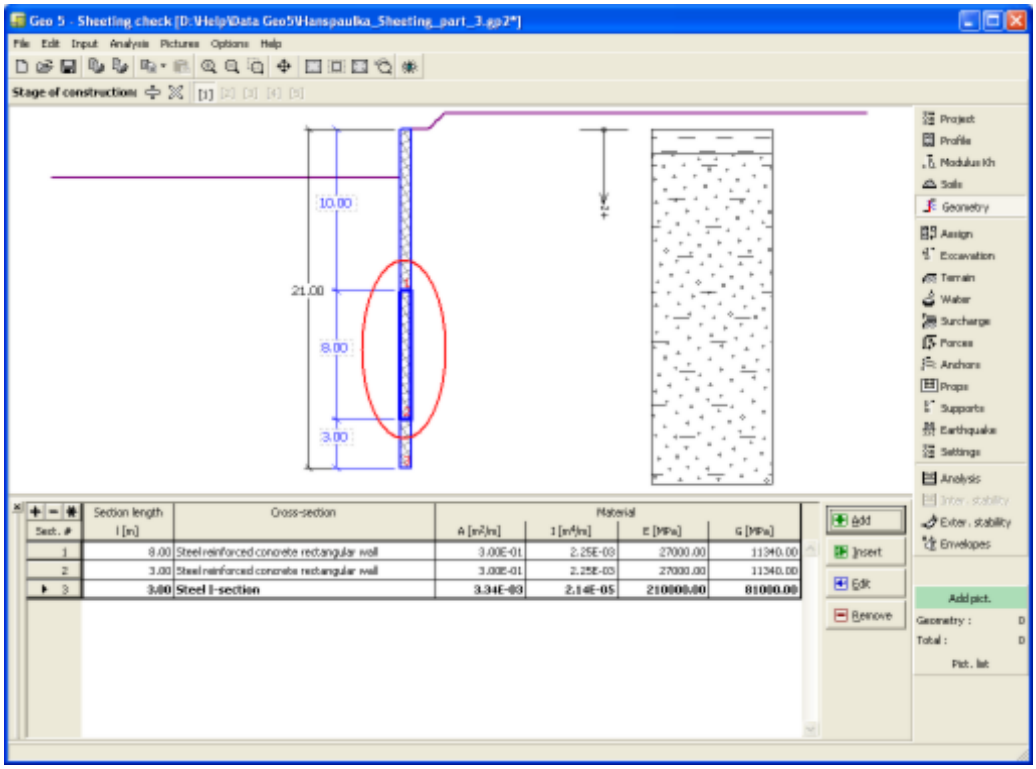
Frame "Soils"

Geometry

The "Geometry" frame contains a table with a list of inputted structural sections forming the sheeting wall. For each section the table stores its cross-sectional characteristics (A – area, I – Moment of inertia) and material characteristics (E – Modulus of elasticity, G – Shear modulus – these variables are always expressed with respect to 1m of a construction length). Adding (editing) sections is performed in the "New section (Edit section)" dialogue window.

The program allows for adding (inserting) another section in between two already existing sections of a structure. Inserting a new section is performed in the "Insert section" dialogue window that complies with the "New section" dialogue window. The inserted section is ordered such as to precede the currently selected section of a structure.

The inputted sections can be further edited on the desktop with the help of active objects – double clicking on a structure opens a dialog window with a given section.



Frame "Geometry"

Adding and editing section

The "New section (Edit, section, Insert section)" dialog window contains the following items:

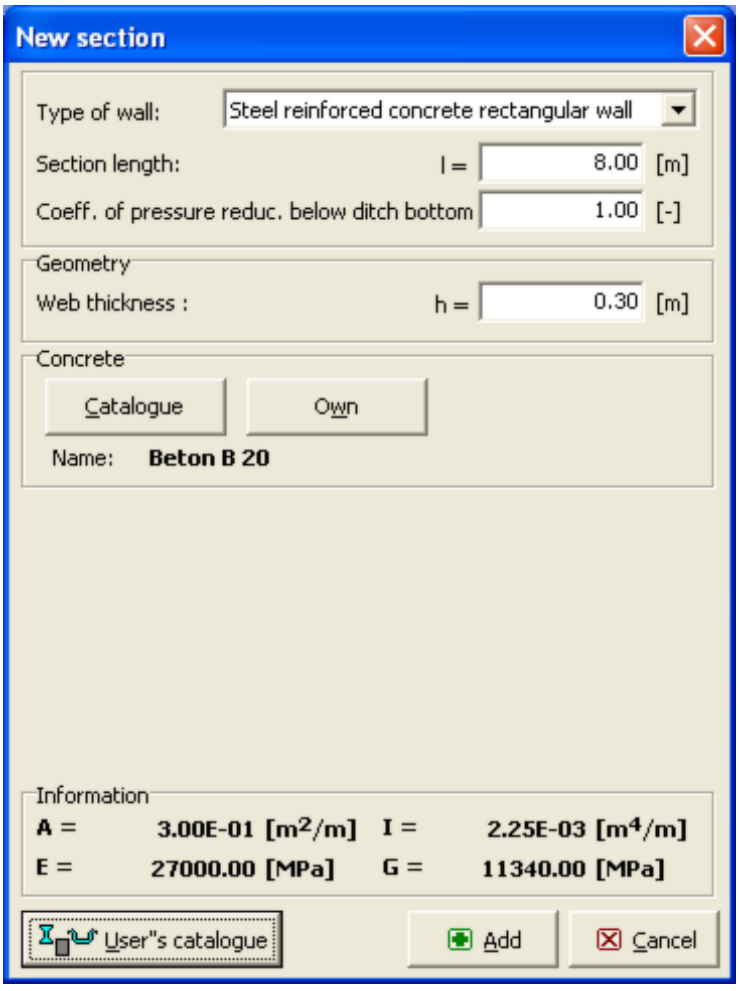
Type of wall
- combo list containing individual types of walls to create a sheeting (pile wall, reinforced concrete rectangular wall, sheet pile wall, steel I cross-section, or own generation of desired cross-sectional characteristics

Section length
- use input field to specify length of sheeting wall or length of a given section of a structure, respectively

Coeff. of reduc. of earth press. below ditch bottom
- coefficient allowing for computation of braced sheeting (for classical sheeting is set equal to one)

Geometry	• contains information about geometry for a given structural variant (e.g., for pile wall it contains pile cross-section and spacing of piles, for reinforced concrete wall its thickness, etc.)
Profile	• contains information about profile for the selected structural variant "steel I cross-section" (buttons "Catalogue" and "Edit" open the "Rolled section steel profiles", which contains a list of these cross-sections)
Material	• contains information about material of a given structural variant (e.g., for pile wall it contains a catalogue of materials to select the type of concrete, for sheet pile wall the elastic modulus, etc.)
Information	• contains overview of cross-sectional characteristics of the inputted cross-section – area and moment of inertia are always evaluated for 1m of length in out of plane direction

The "User catalogue" button in the bottom part of the window opens the "User catalogue" dialogue window.



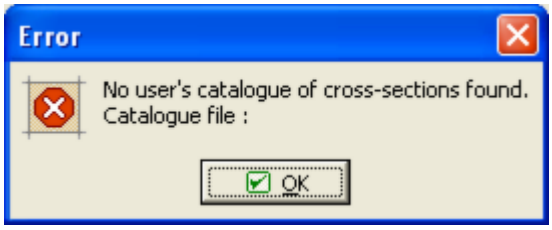
Dialogue window "New section"

User catalogue

The user catalogue allows the user to define and store own cross-sections and their characteristics that appear in the construction of a wall. At first use of the catalogue (has not been yet created) the program prompts a warning message that no catalogue was found. Then, pressing the button "OK" opens the "Save as" dialogue window that allows for entering

the catalogue name and saving it into a specified location by pressing the "Save" button (by default a folder used for saving the project data is assumed).

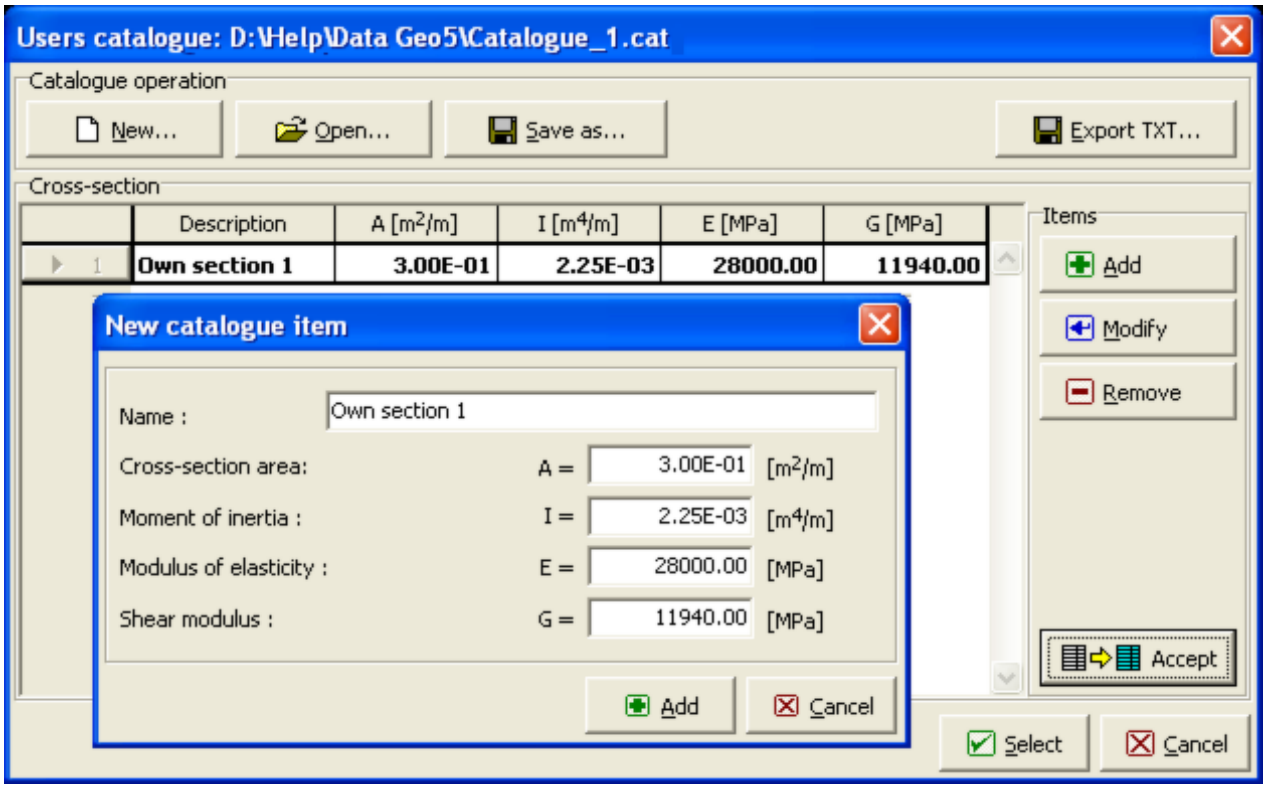
The program allows the user to create more than one catalogue. The next catalogue is created by pressing the "New" button – the program asks, whether the current catalogue should be replaced (the currently loaded catalogue is not deleted!) and saves the new catalogue under a new name. The "Open" button allows for loading an arbitrary user catalogue and by pressing the "Save as" button for saving it under a different name.



Dialogue window at first use user catalogue of cross-sections

The "User catalogue" dialogue window contains a table listing the user defined cross-sections. The "Add item" button opens the "New catalogue item" dialogue window that allows for specifying and subsequent saving of characteristics of a new cross-section into the catalogue. Buttons "Edit item" and "Remove item" serve to edit individual items in the table.

The "Accept current" button accepts the current cross-sectional characteristics of a cross-section specified in the "New section" dialogue window and opens the "New catalogue item" dialogue window that allows for modifying and saving the current cross-section.

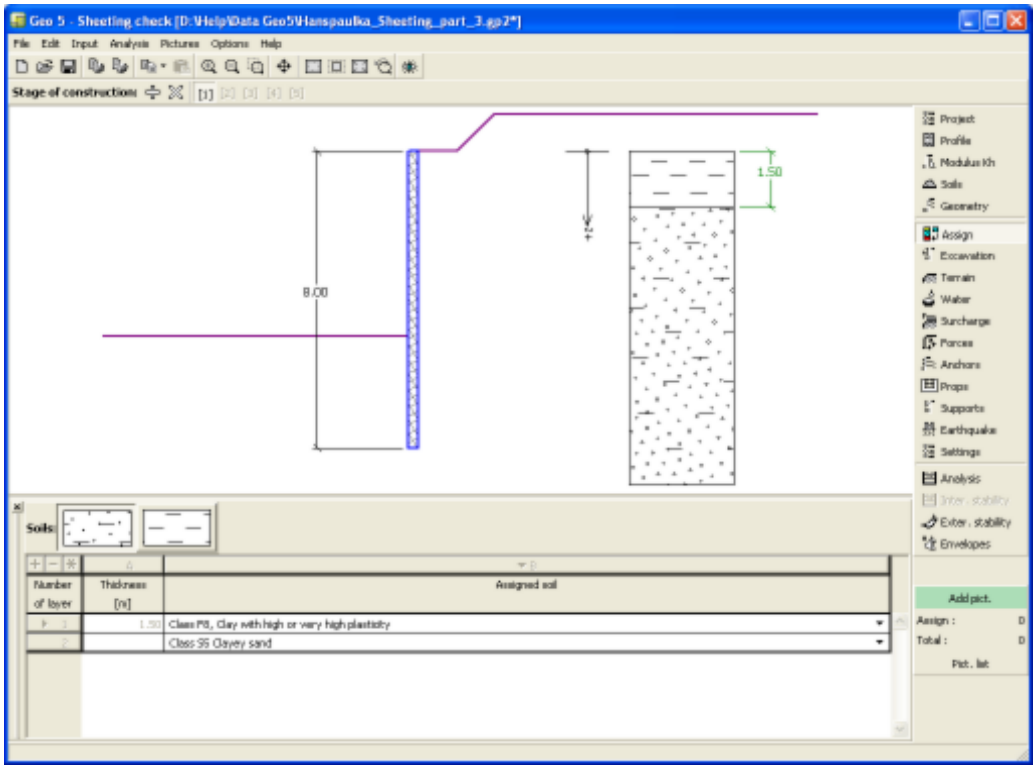


Dialogue windows "User catalogue" and "New catalogue item"

Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

Procedure to assign soil into a layer is described in details herein.

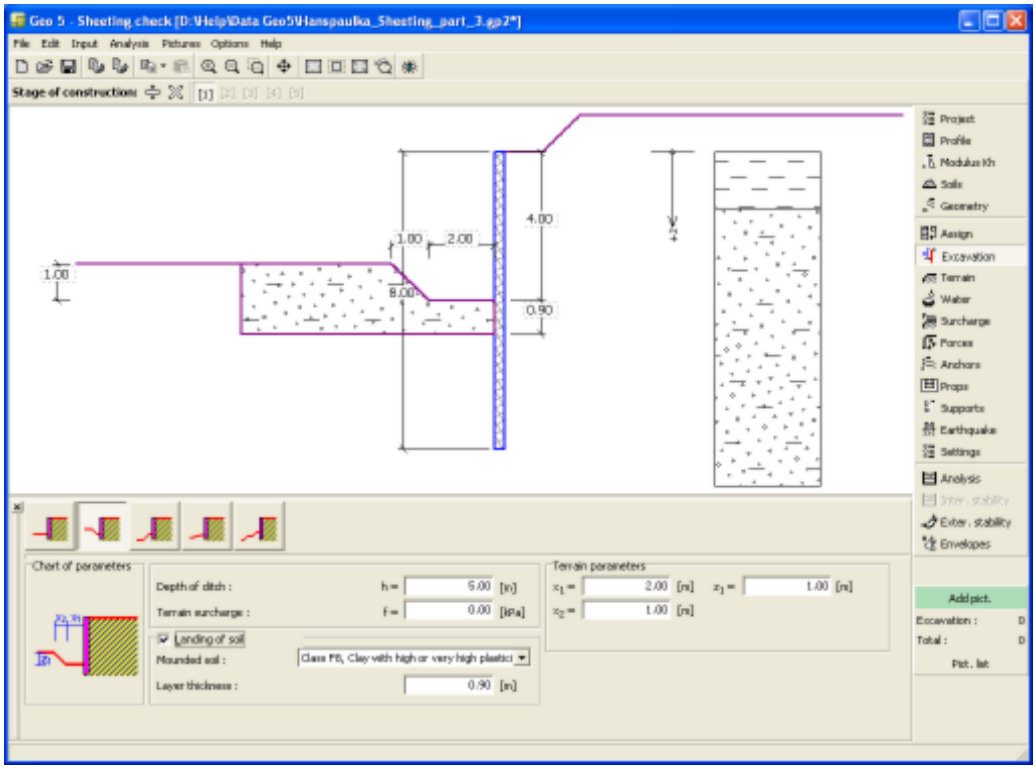


Frame "Assign"

Excavation

The "Excavation" frame serves to input the depth of a construction ditch and by pressing the button to select the shape of the ditch base. The selected shape with a graphic hint " Parameter chart" appears in the left part of the frame. The dimensions of a structure can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The frame also allows for specifying surcharge acting on the ditch base or a thickness of made-up ground of new soil below the ditch base (the soil can be selected from a combo list containing soils inputted in the regime "Soils"). When introducing the made-up ground with brace sheeting it is assumed that there is a sheeted structure in the location of made-up ground, i.e., all pressures are acting on the entire width of a structure as above the construction ditch base.



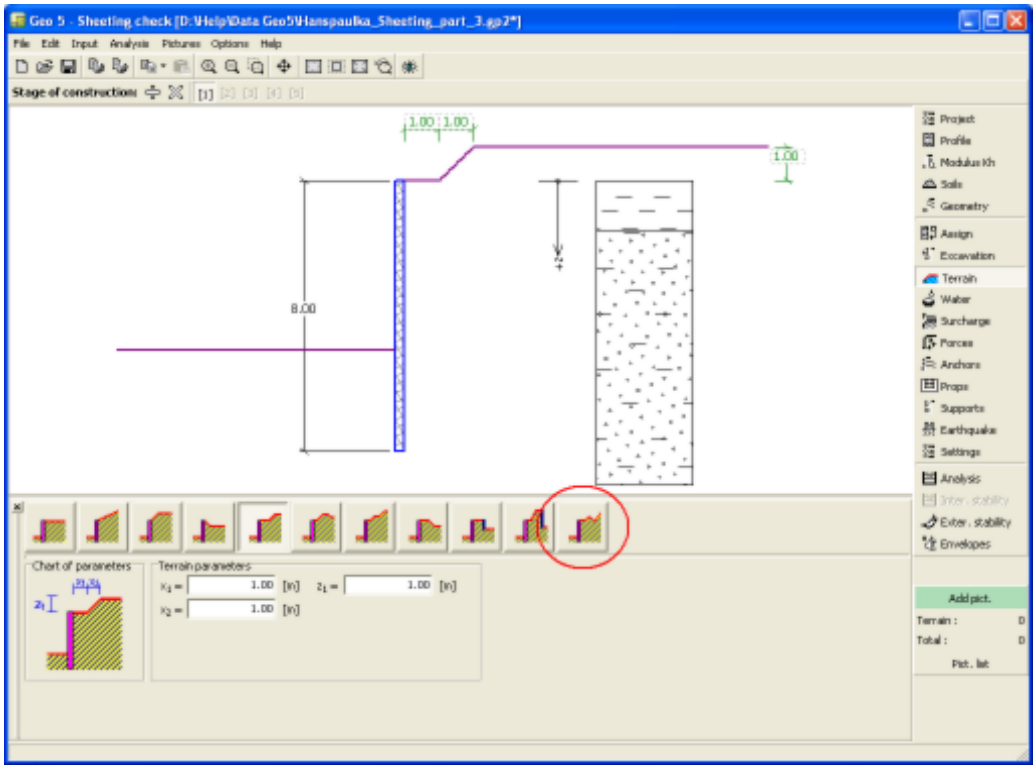
Frame "Excavation"

Terrain

The "Terrain" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0, 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the hint, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

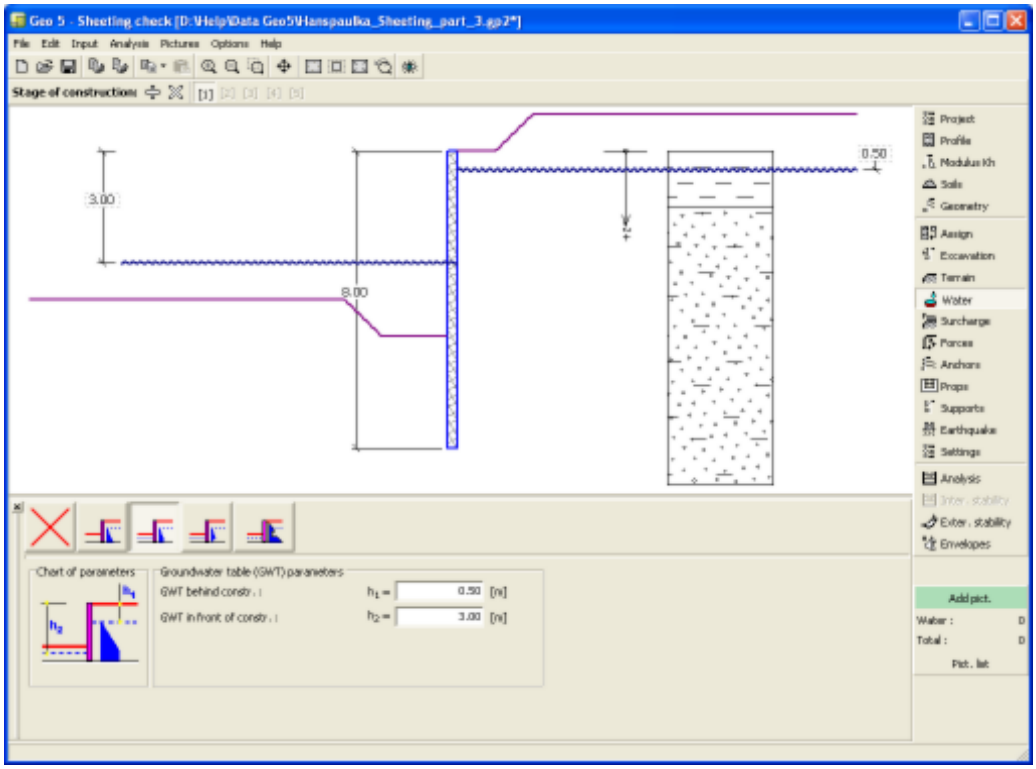
The "Water" allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint (“Parameter chart”) of inputted values is displayed in the left part of the frame. Water parameters (h1, h2...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option is a manual input of pore pressure both in front and behind the structure. Two tab sheets "In front of structure" and "Behind structure" appear with tables. The table is filled with values of pore pressure in front, or behind the structure at a depth of “z” (z-axis).

The ground water table can also be specified above the structure or earth profile, respectively – in such a case the depth of water is inputted with negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the hint chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



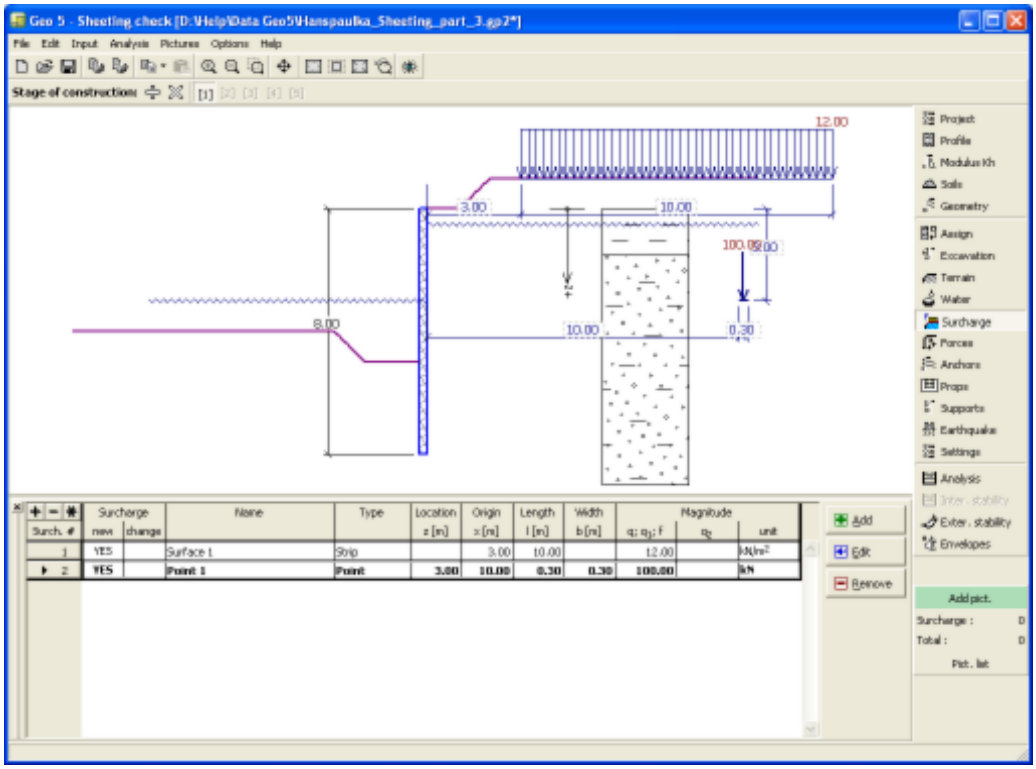
Frame "Water"

Surcharge

The "Surcharge" frame contains a table with a list of inputted surcharges. Adding (editing) surcharge is performed in the "New (edit) surcharge" dialogue window. The inputted surchages can be edited on the desktop with the help of active dimensions or active objects, respectively.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. Providing the surcharge is found off the terrain the computer prompts an error message.

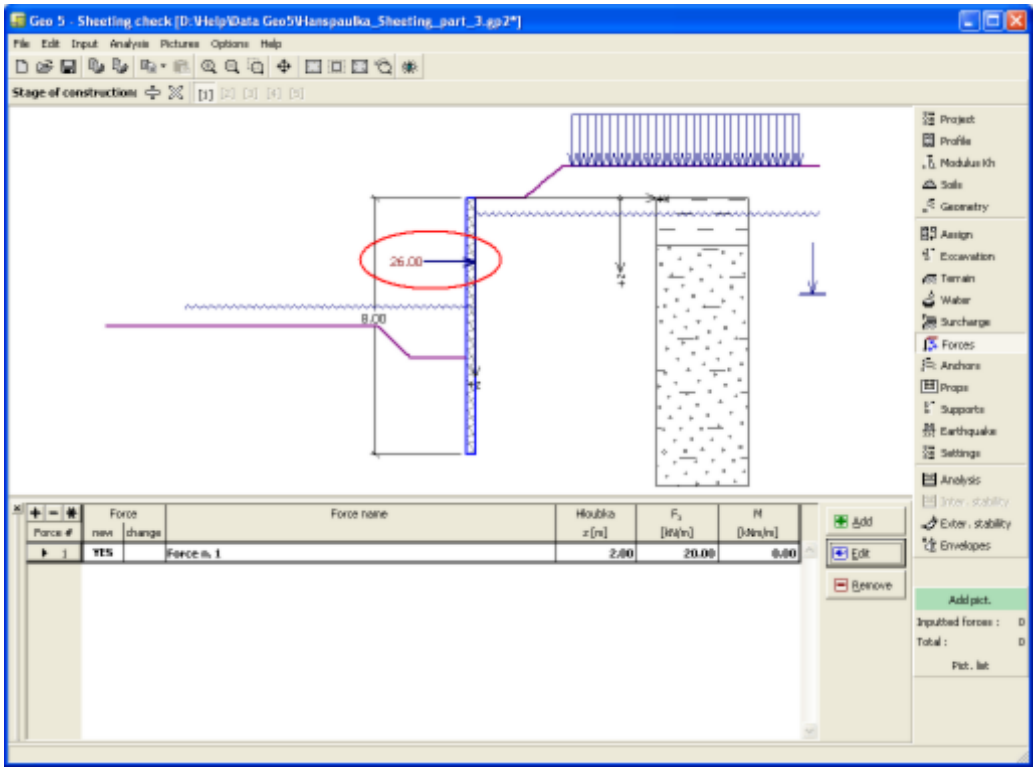
Analysis of earth pressures due to surcharges is described in the theoretical part of the hint, chapter "Influence of surcharge".



Frame "Surcharge"

Forces

The "Forces" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "New force (edit force)" dialogue window. The inputted forces can also be edited on the desktop with the help of active objects.



Frame "Forces"

Anchors

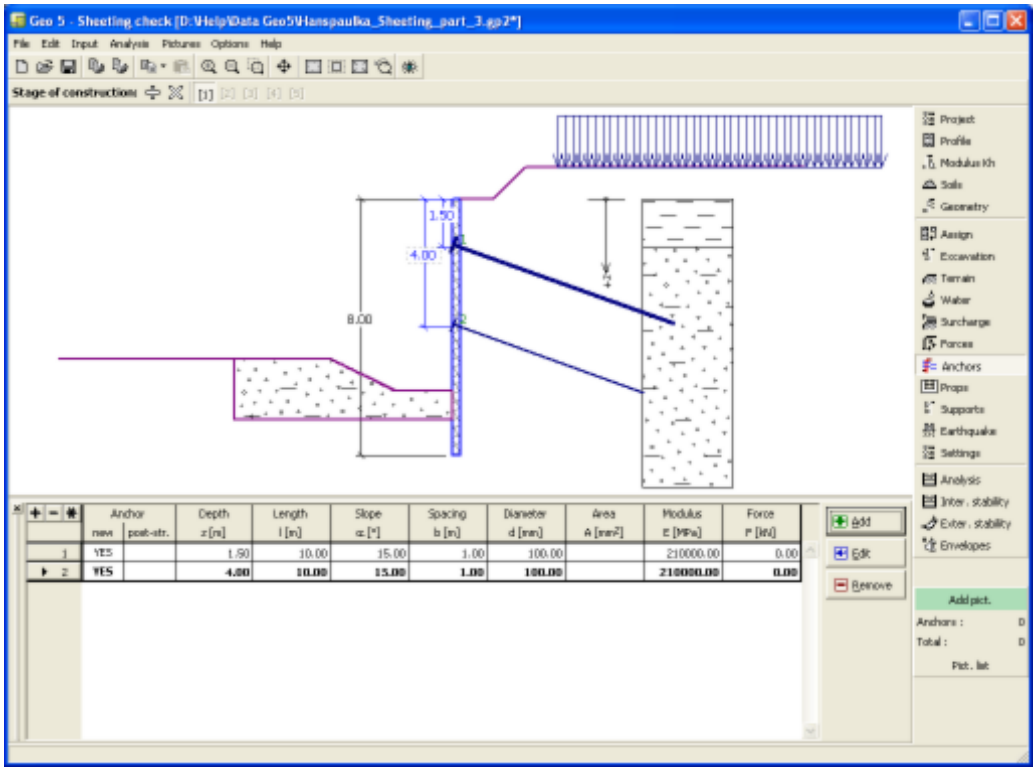
The "Anchors" frame contains a table with a list of inputted anchors. Adding (editng)

anchors is performed in the "New anchor (Edit anchor)" dialogue window. The inpuuted anchors can be edited on the desktop with the help of active objects.

You are asked to input the anchor location, its length and inclination, pre-stress force and parameters needed to determine the anchor stiffness (cross-sectional area, modulus of elasticity). The anchor is introduced automatically on already deformed structure (obtained from the previous stage of construction).

The anchor stiffness becomes effective in subsequent stages of construction. In subsequent stages the anchor can no longer be edited. The only action available is the change of anchor pre-stress force.

Note: The program does not check the anchor bearing capacity against breakage.



Frame "Anchors"

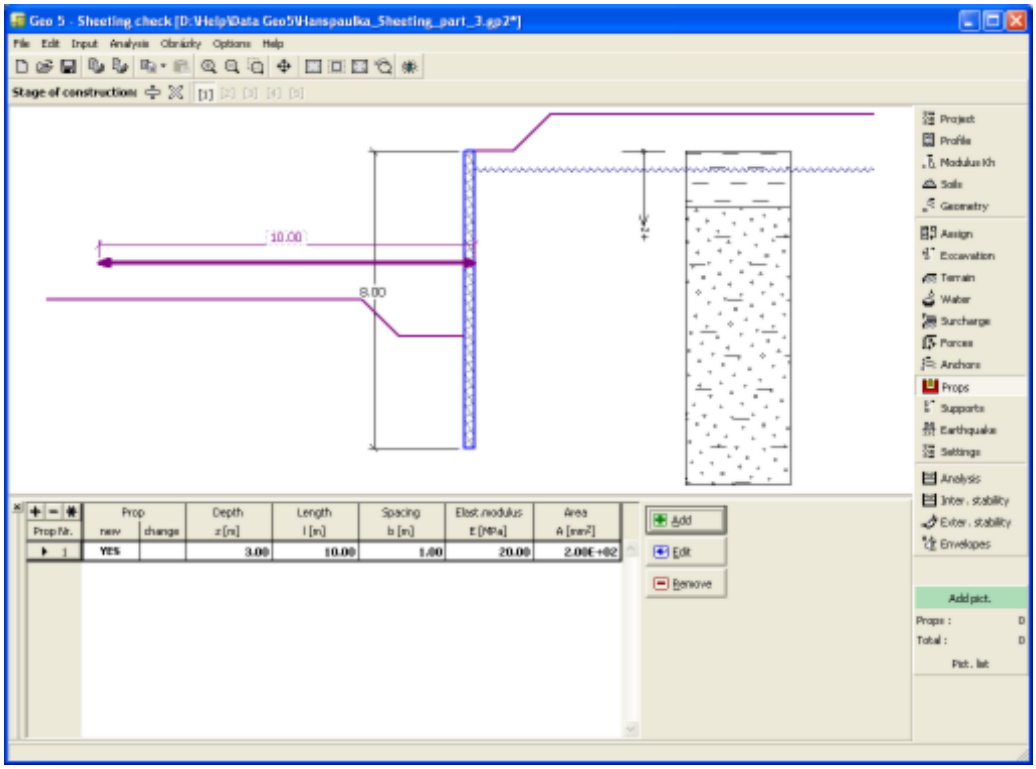
Props

The "Props" frame contains a table with a list inputted props. Adding (editing) props is performed in the "New prop (Edit prop)" dialogue window. The inputted props can also be edited on the desktop with the help of active dimensions or active objects, respectively.

You are asked to input the prop location, its length and parameters needed to determine the prop stiffness (cross-sectional area, modulus of elasticity).

The prop is introduced automatically on already deformed structure (obtained from the previous stage of construction). In subsequent stages the props can no longer be edited. The only action available is the change of prop stiffness.

Note: The program does not check the prop bearing capacity neither for compression nor for buckling.

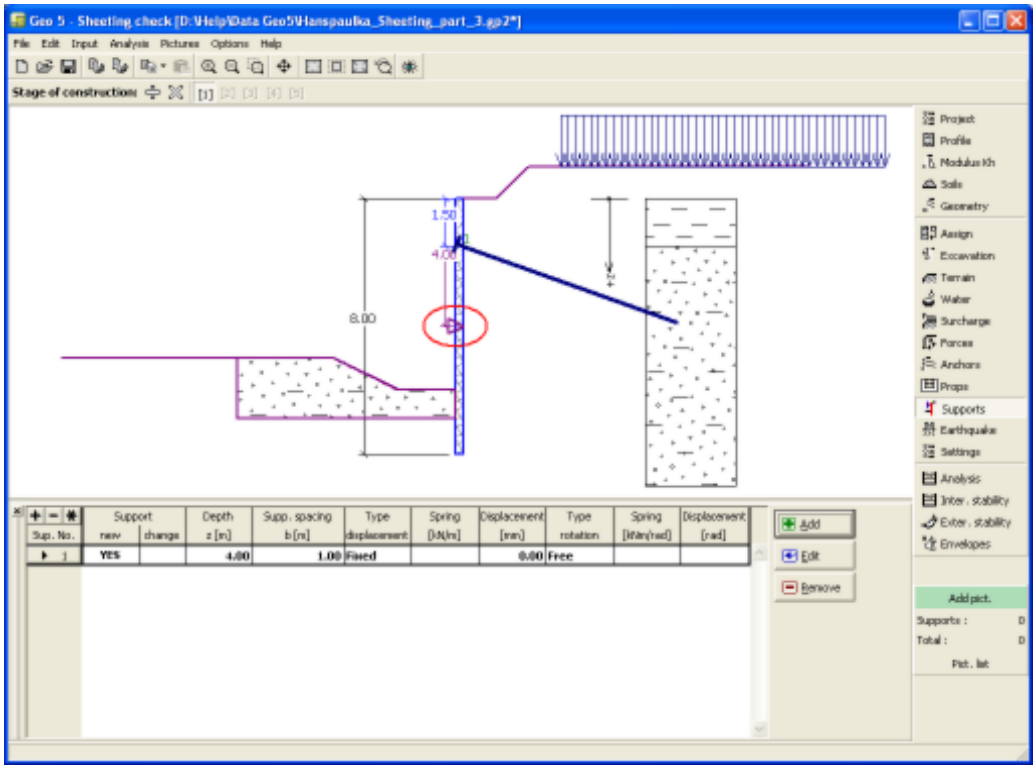


Frame "Props"

Supports

The "Supports" frame contains a table with a list of inputted supports. Adding (editing) supports is performed in the "New support (Edit support)" dialogue window. The inputted supports can also be edited on the desktop with the help of active dimensions or active objects, respectively.

You are asked to specify the support type (free, fixed, and spring) and its location. The support is inputted automatically on already deformed structure (obtained from the previous stage of construction). In subsequent stages the supports can no longer be edited. The only action available is the introduction of prescribed (forced) displacement in a support.

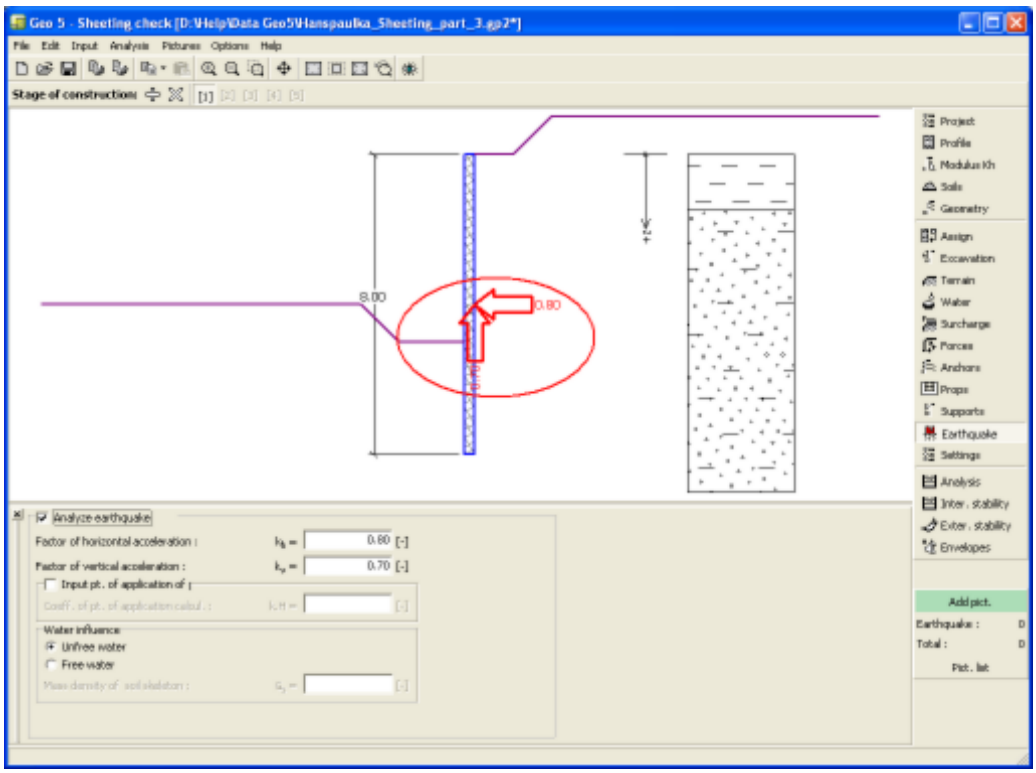


Frame "Support"

Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of inputted earthquake effects are displayed on the desktop.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the hint in chapter "Influence earthquake".



Frame "Earthquake"

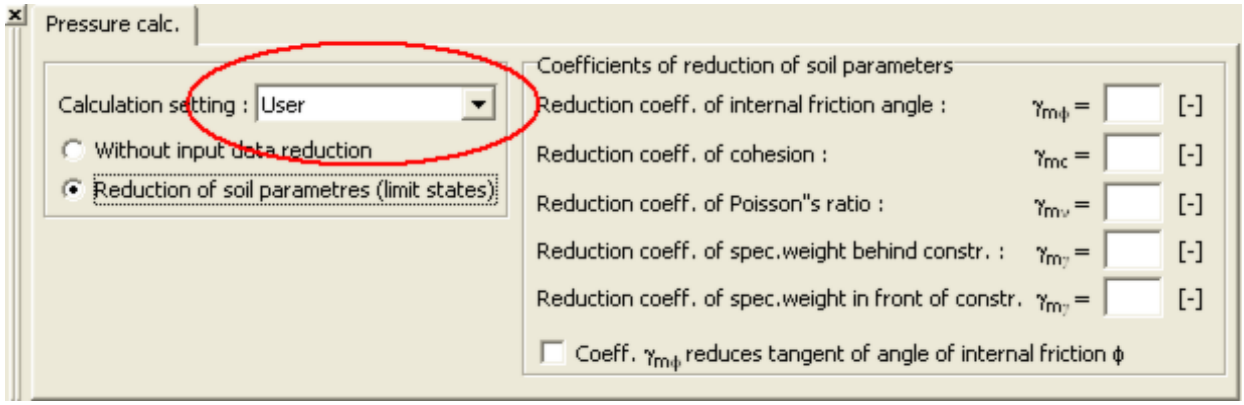
Setting

The "Setting" frame contains basic settings for the analysis of earth pressures. The program offers pre-setting for different countries (Czech Republic, Germany, France,...) and " Standard setting" recommended by the authors of the program for countries no included in the list. While changing settings in the combo list the values of coefficients of reduction of soil parameters in corresponding windows are changed.

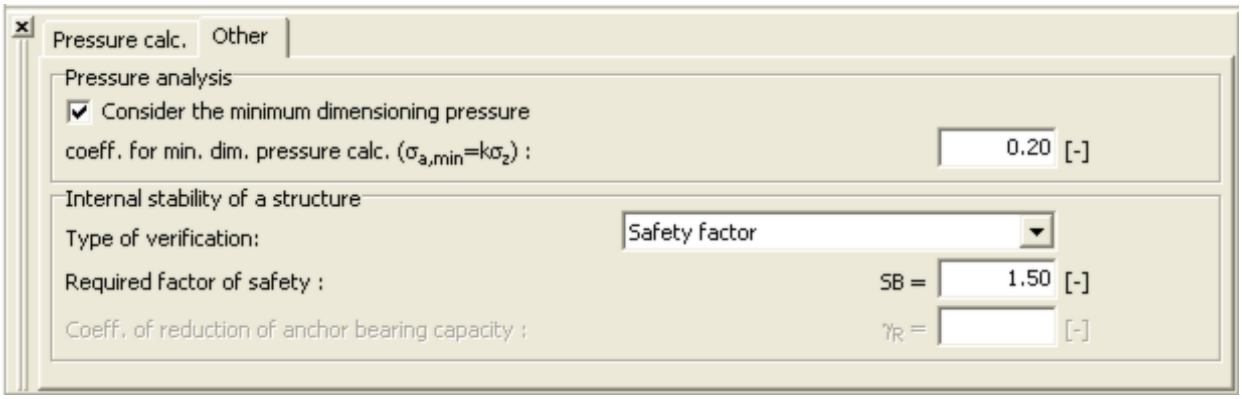
An arbitrary analysis setting is available with the option "User setting". Selecting the option "Reduce soil parameters" allows for specifying in input fields individual values of the coefficients of reduction of soil parameters (e.g., recommended values according to EC7).

The "Other" tab sheet contains setting for minimum dimensioning pressure and setting of parameters of internal stability of a structure.

The program "Sheeting check" is useful particularly when modeling real behavior of a structure – consequently we recommend the analysis to be performed without reduction of input characteristics of soils ("Standard setting").


Pressure calc.
Calculation setting : User
Without input data reduction
Reduction of soil parametres (limit states)
Coefficients of reduction of soil parameters
Reduction coeff. of internal friction angle : γmφ = [-]
Reduction coeff. of cohesion : γmc = [-]
Reduction coeff. of Poisson"s ratio : γmν = [-]
Reduction coeff. of spec.weight behind constr. : γmγ = [-]
Reduction coeff. of spec.weight in front of constr. γmγ = [-]
Coeff. γmφ reduces tangent of angle of internal friction φ

Frame "Setting" - tab sheet "Pressure calculation"


Pressure calc.
Other
Pressure analysis
Consider the minimum dimensioning pressure
coeff. for min. dim. pressure calc. (σa,min=kσz) : 0.20 [-]
Internal stability of a structure
Type of verification: Safety factor
Required factor of safety : SB = 1.50 [-]
Coeff. of reduction of anchor bearing capacity : γR = [-]

Frame "Setting" - tab sheet "Other"

Analysis

The frame "Analysis" displays the analysis results. Switching to this regime automatically runs the analysis. The frame contains three buttons to show the analysis results:

- K_h + pressures

Variation of the modulus of subsoil reaction is displayed in the left part of the desktop (by default a blue color with hatching) is assumed. Referring to the method of depending pressures some of the springs (values of modules of subsoil reaction) are removed (spring stiffness set equal to zero) from the analysis. The analysis may fail to converge providing the critical (limit) state developed both in front and behind the structure and there is not enough constrains available (anchors, supports). The program exists without finding a solution. An error message appears in the bottom part of the frame – such a case calls for modification in problem input – e.g., add an anchor, change a depth of excavation, improve soil parameters, etc.
Some construction stages display (by default a yellow dotted line is assumed) deformation at the onset of mobilization of the earth pressure at rest – this is a complementary information showing plastic deformation of a structure.

Distributions of limiting pressures (by default a green dashed line is assumed) are presented in the right part of the window (passive pressure, pressure at rest and active pressure). The actual pressure acting on a structure is plotted in a solid blue line.

Both deformed (by default a solid red color is assumed) and undeformed structure appears in the right part of the desktop. Forces and displacements developed in anchors, supports and props are also shown.

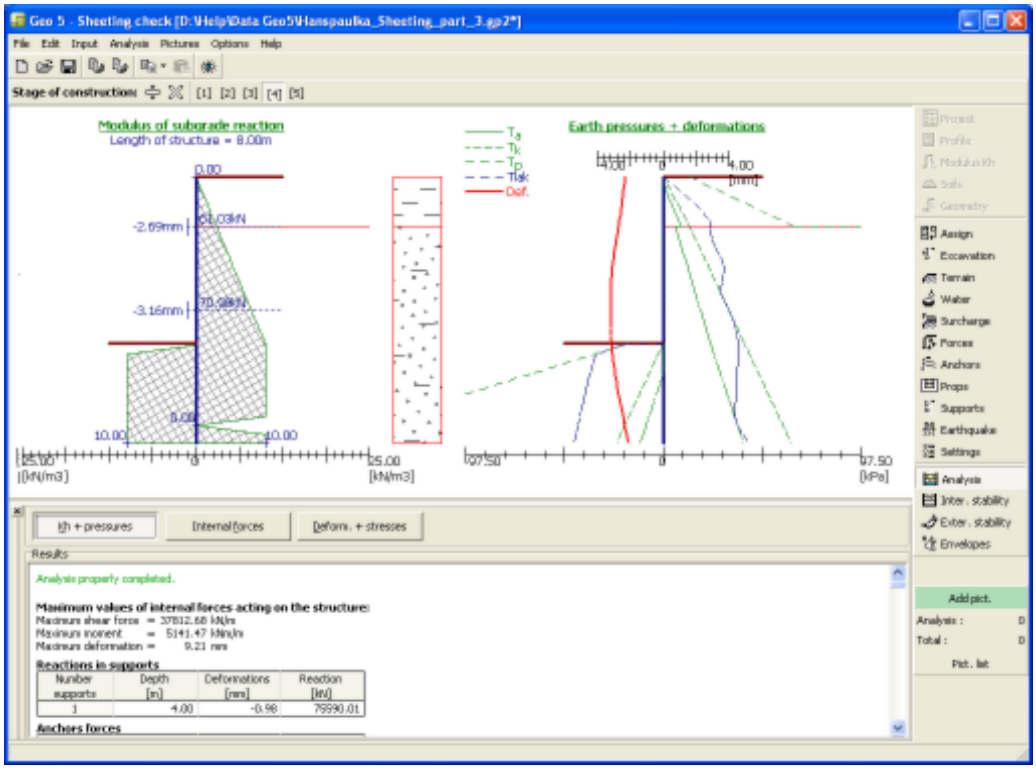
- Internal forces

Plot of a structure together with forces acting in anchors, reactions and deformations of supports and props appear in the left part of the desktop. Distributions of bending moment and shear force are then plotted on the right.

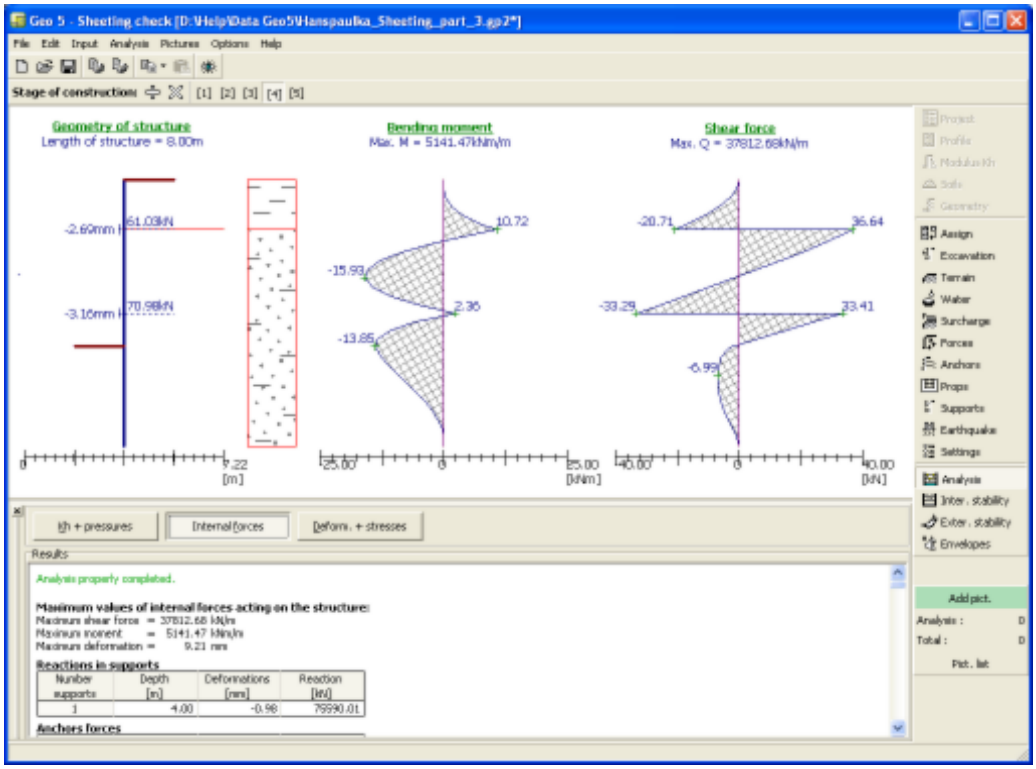
- Deformations + Stresses

Plot of a structure together with forces acting in anchors, reactions and deformations of supports and props appear in the left part of the desktop. The deformed shape of a structure together with overall pressure acting on a structure is then plotted on the right.

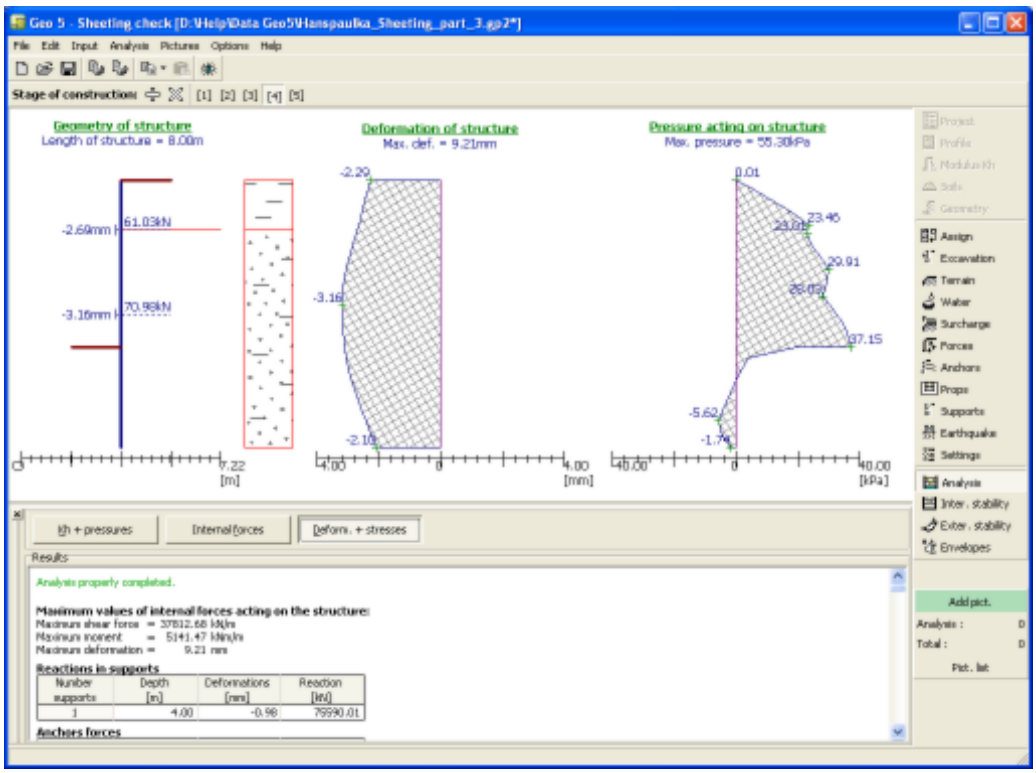
Providing the modulus of subsoil reaction is found by iteration it is necessary to check the course of iteration in the dialogue window. Details are provided in the theoretical part of the hint "Modulus of subsoil reaction determined by iteration".



Frame "Analysis" - modulus of susoil reaction, earth pressures and deformations



Frame "Analysis" - bending moment and shear force



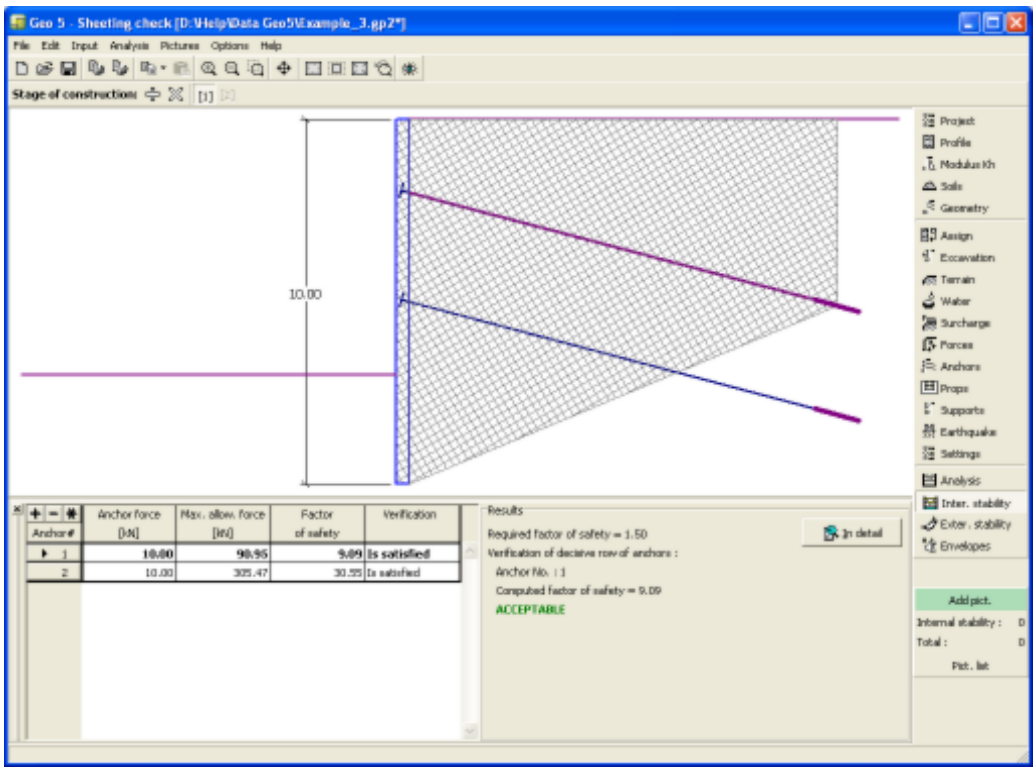
Frame "Analysis" - deformation and pressure acting on structure

Internal stability

This frame serves to check the internal stability of anchors – the frame is therefore accessible only in stages, in which the anchors are introduced. For each row of anchors the table shows inputted anchor forces and the maximum allowable forces in each anchor. Overall check for the most stressed row of anchors is displayed in the right part of the frame.

The verification procedure depends on "Setting" – either based factors of safety or according

to the theory of limit states. The solution procedure is described herein.

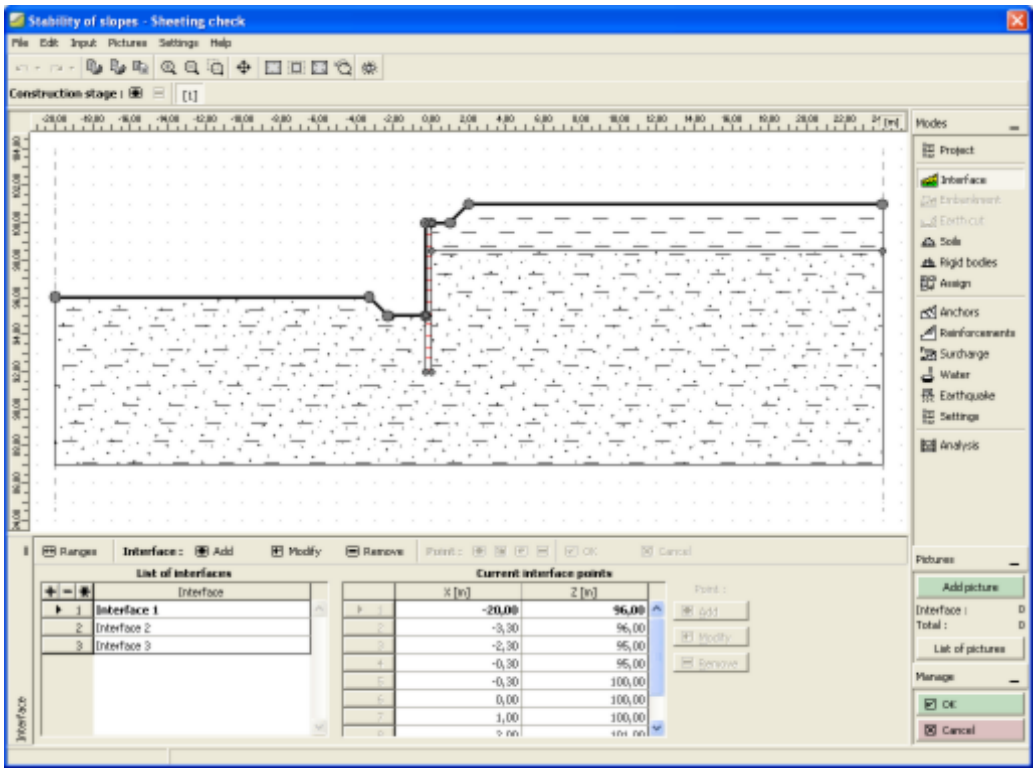


Frame "Internal stability"

External stability

Pressing the "External stability" button launches the "Slope stability" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "Slope stability" is installed.

After completing all analyses press the "OK" button to leave the program – all data are then carried over to the analysis protocol of the "Sheeting verification" program.

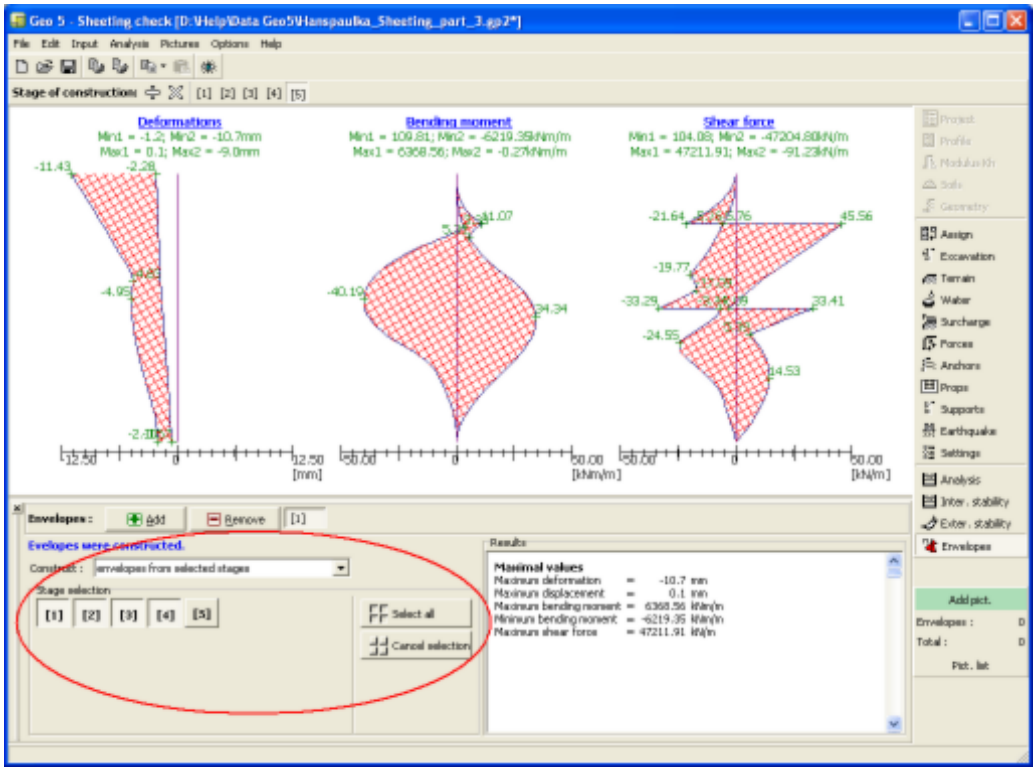


Frame "External stability"

Envelopes

The frame "Envelopes" allows for displaying an envelope of internal forces and displacements from all analyses (stages of constructions). By default the envelope is constructed from the results from all construction stages. It can, however, be created only from the selected stages (pressing buttons selects the constructions stages that are used to generate the current envelope). The program makes possible to construct more envelopes with various combinations.

The maximum internal forces and displacements are displayed in the right part of the frame.



Frame "Envelopes"

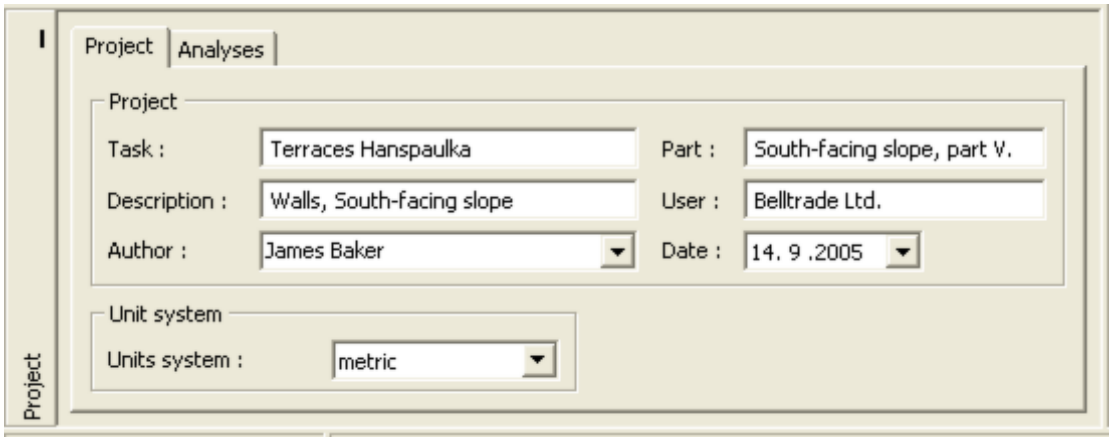
Program Slope Stability

Project

The "Project" frame is used to input the basic project data and to specify the overall setting of the analysis run. The "Project" tab sheet contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc.

The "Project" tab sheet also allows the user to switch analysis units (metric / imperial).

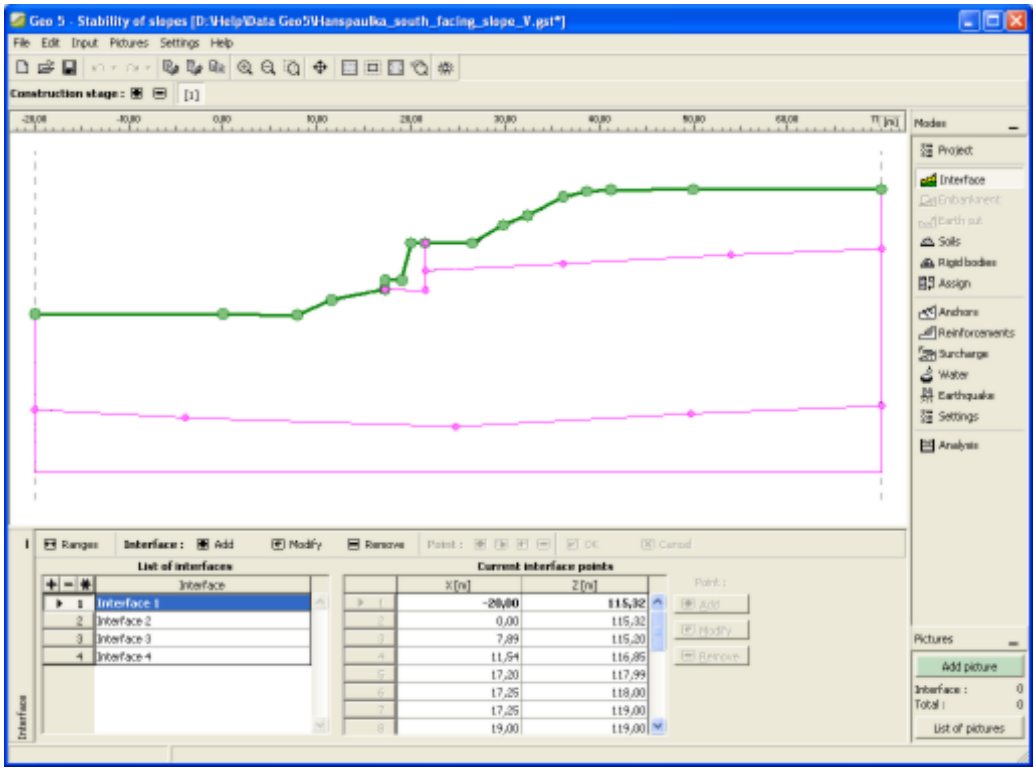
The "Analysis" tab sheet is used to specify whether the analysis is performed in total or effective parameters.



Frame "Project" - tab sheet "Project"

Interface

The "Interface" frame serves to introduce individual soil interfaces into the soil body. Detailed description how to deal with interfaces id described herein.

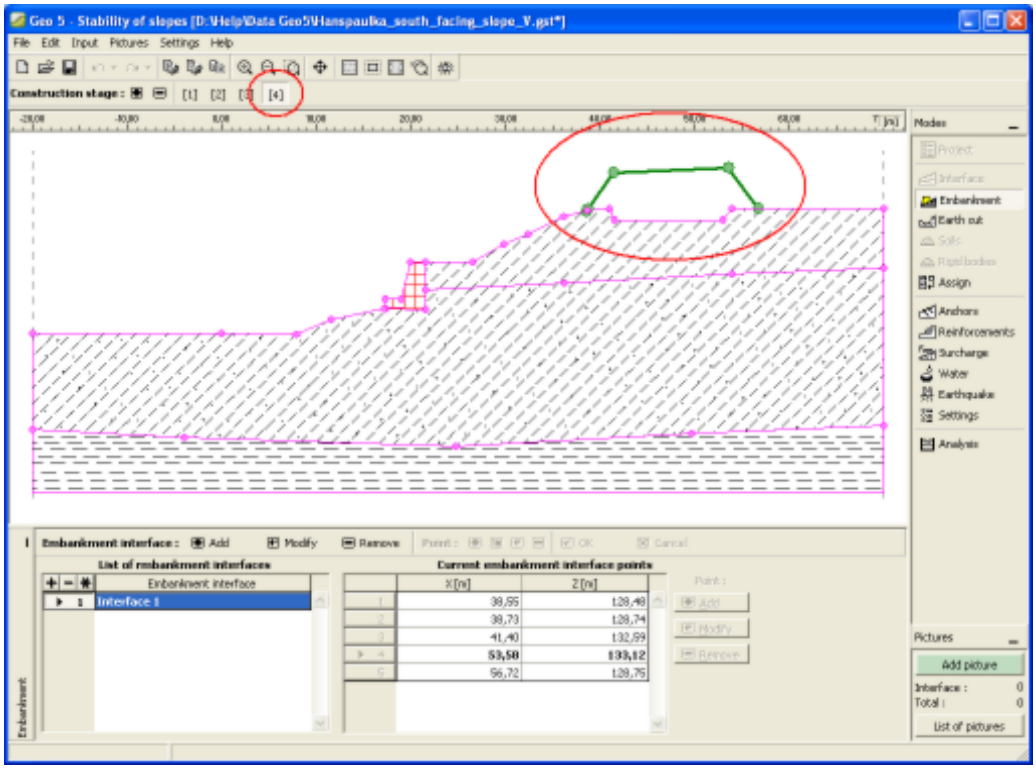


Frame "Interface"

Embankment

The "Embankment" frame allows for inputting interfaces to create an embankment above the current terrain. The frame contains a table with a list of interfaces forming the embankment. A table listing the points of currently selected interface of the embankment is displayed in the mid section of the frame. Inputting an embankment interface follows the same steps as used for standard interfaces.

An embankment cannot be specified in the first stage of construction. An embankment cannot be built if there is an earth cut already specified in a given stage - in such a case either a new stage of construction must be introduced for embankment input or the existing open cut must be first removed.



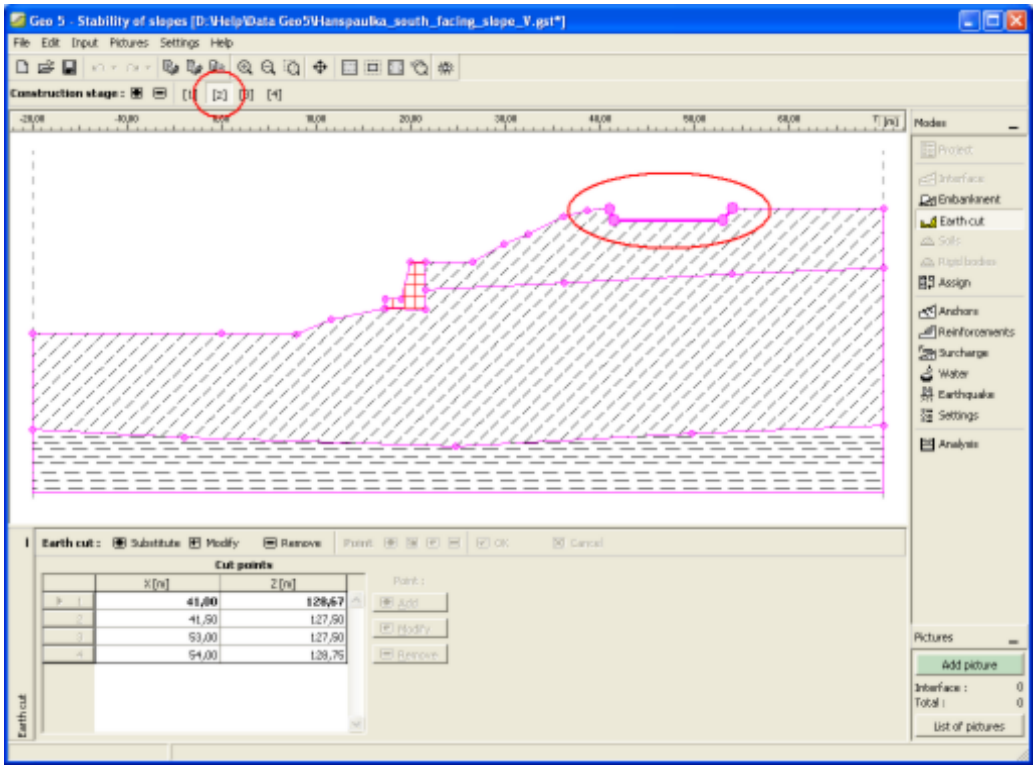
Frame "Embankment"

Earth cut

The "Earth cut" frame serves to specify the shape of an open cut. This function allows for modifying the terrain profile within a given stage of construction. Several earth cuts can be introduced at the same time. In such a case some of the lines in the cut appear partially above the terrain.

A table listing individual interface points is displayed in the left part of the frame. Inputting an earth cut interface follows the same steps as used for standard interfaces.

An open cut cannot be specified in the first stage of construction. An earth cut cannot be built if there is an embankment already specified in a given stage - in such a case either a new stage of construction must be introduced for earth cut input or the existing embankment must be first removed.



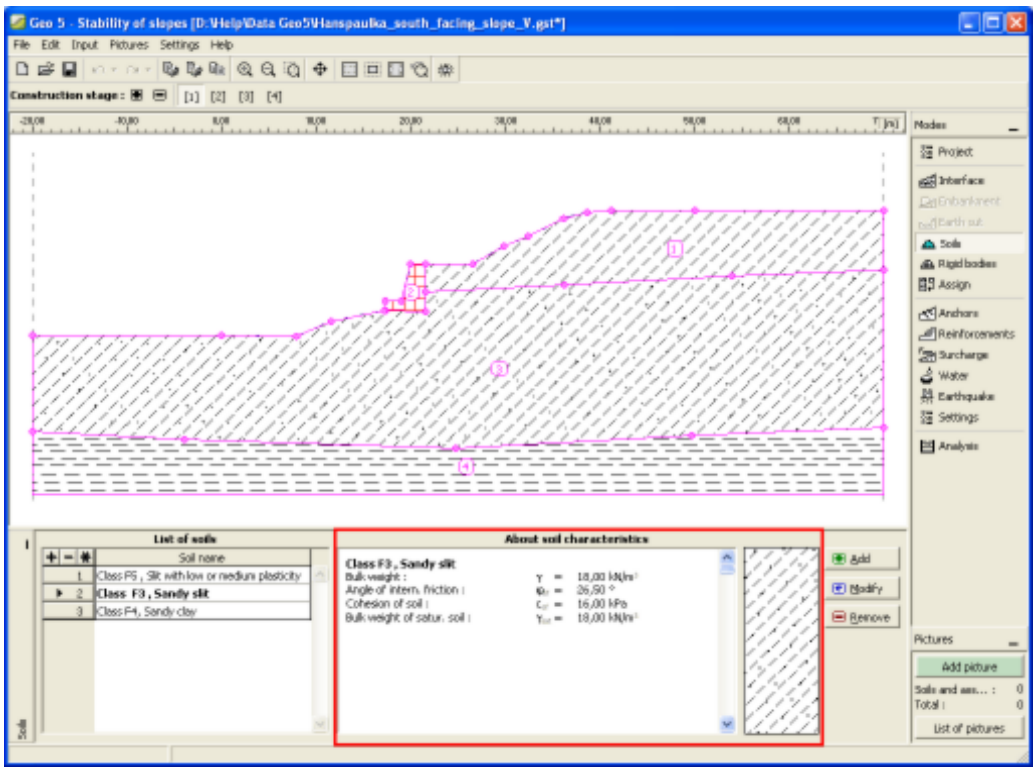
Frame "Earth cut"

Soils

The "Soils" frame contains a table with a list of inputted soils. The table also provides information about currently selected soil displayed in the right part of the frame.

Adding (editing) a soil is performed in the "Add (edit) soil" dialogue window.

The soil characteristics are specified in the program "Slope stability". These characteristics are further specified in chapters: "Uplift pressure" and "Foliation". An input of parameters further depends on the selected type of analysis (effective / total stress state), which is set in the frame "Projekt", tab sheet "Analysis".



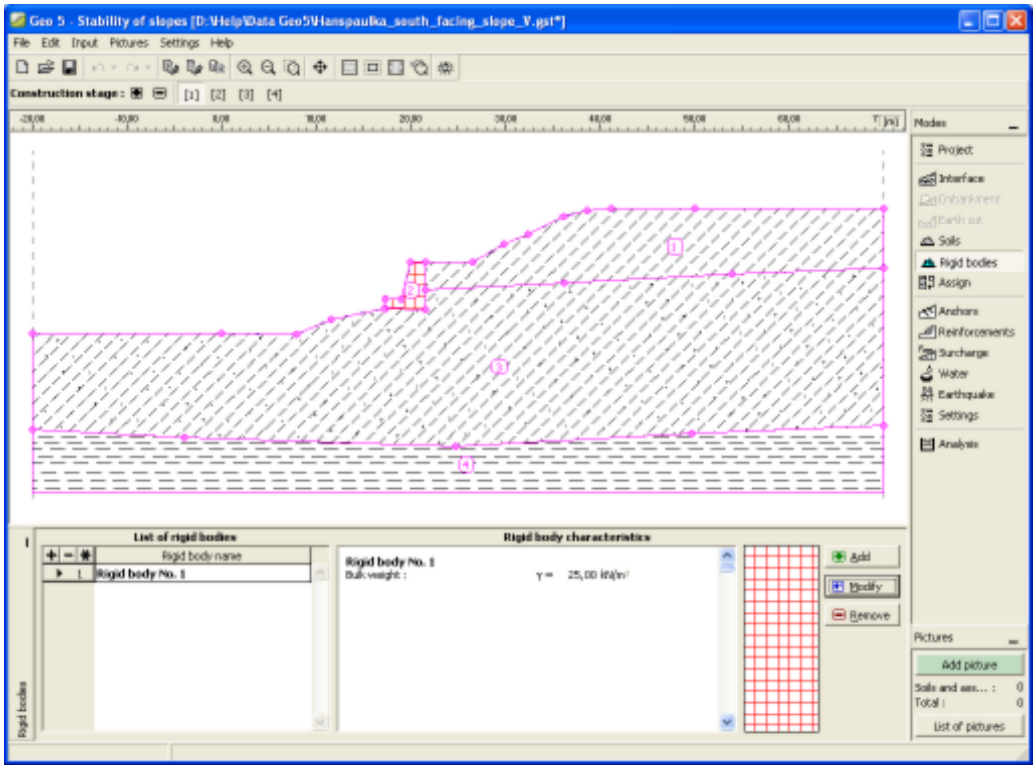
Frame "Soils"

Rigid bodies

The "Rigid bodies" frame contains a table with a list of inputted rigid bodies. The rigid bodies serve to model regions with a high stiffness – e.g., sheeting structures or rock subgrade. This table also provides information about the currently selected rigid body displayed in the right part of the frame.

Adding (editing) rigid bodies is performed in the "Add new rigid body" dialogue window. This window serves to input the unit weight of the rigid body material and to select color and pattern. The rigid bodies are in the frame "Assign" ordered after inputted soils.

Rigid bodies are introduced in the program as regions with high strength so they are not intersected by a potential slip surface. Providing we wish the slip surface to cross a rigid body (e.g., pile wall) it is recommended to model the rigid body as a soil with a cohesion corresponding to pile bearing capacity against slip.

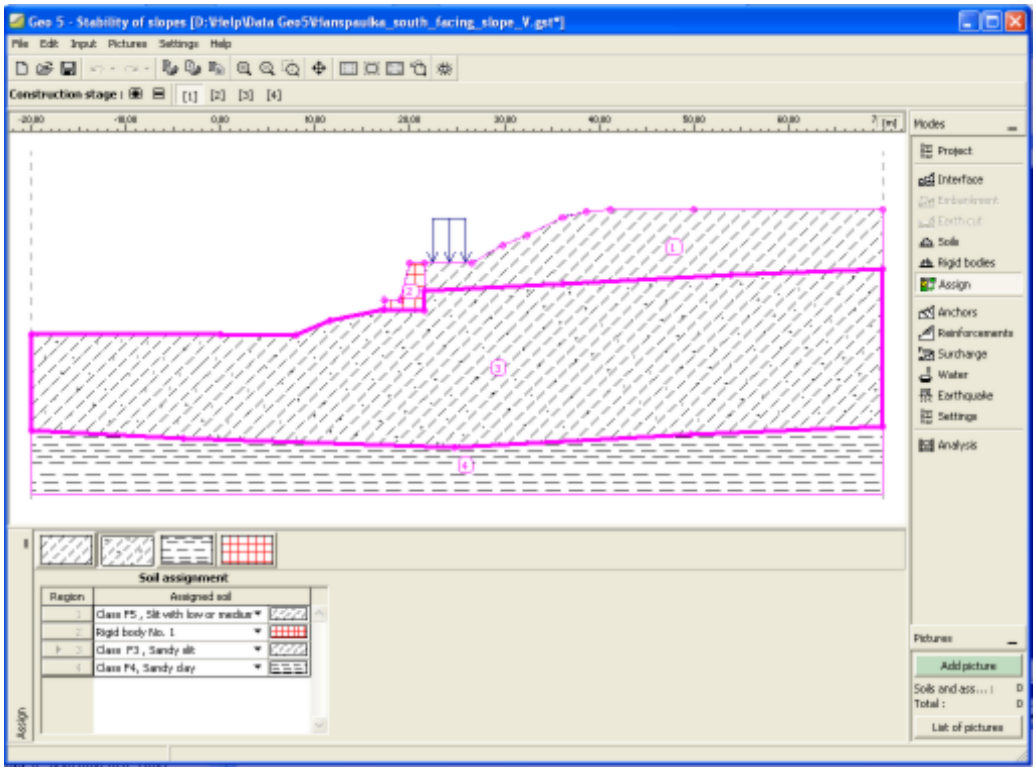


Frame "Rigid bodies"

Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

Procedure to assign soil into a layer is described in details herein.



Frame "Assign"

Anchors

The "Anchors" frame contains a table with a list of inputted anchors. Adding (editing) anchors is performed in the "New anchor (Modify anchor parameters)" dialogue window. The inputted anchors can be edited on the desktop with the help of active objects.

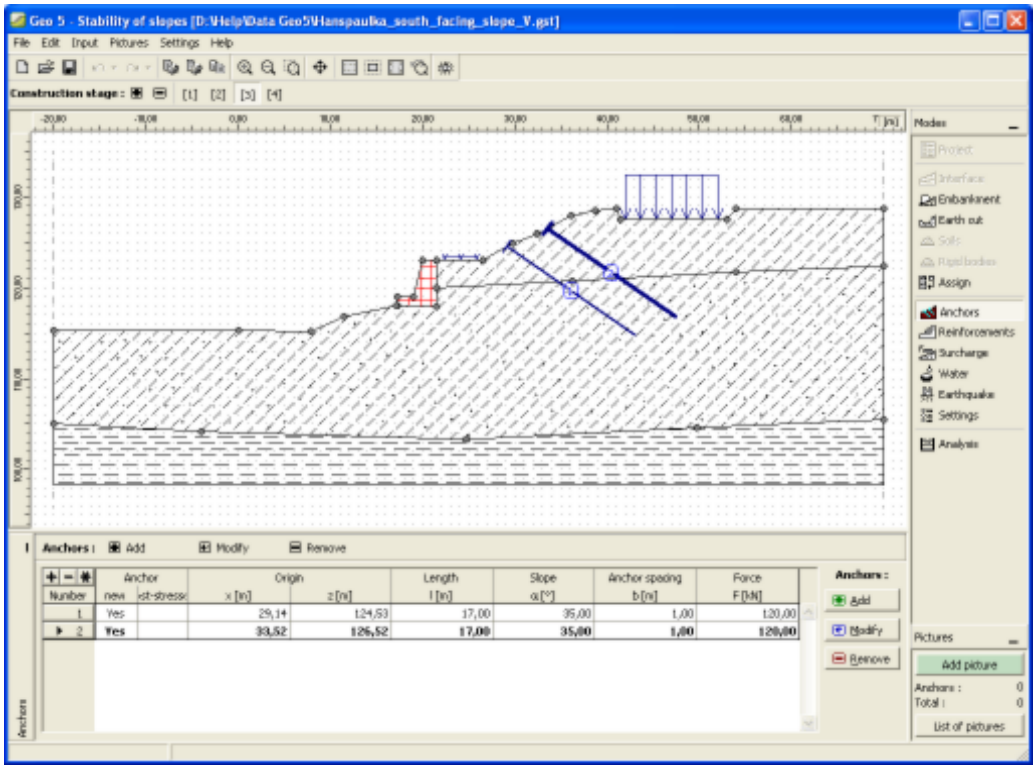
You are asked to input the anchor location (starting point), its length and inclination, anchor spacing and pre-stress force. The anchor starting point is always attached to the terrain. All inputted parameters can be modified in the stage of construction, in which the anchor was introduced. In subsequent stages the program allows only for modifying the anchor pre-stress force (option "Anchor post-stressing").

Anchors can also be introduced with help of a mouse click. Mouse input mode is determined by pressing buttons on the horizontal bar "Anchor". The following modes are available:

- Add — clicking the left mouse button allows for specifying the starting and end point of an anchor. The grid function can be exploited in the input mode. The starting point is always "attached" to the terrain. The coordinates of inputted points are automatically round up to two digits – both input modes (manual, mouse) are therefore identical
- Modify — clicking the left mouse button on already existing anchor opens the "Modify anchor property" dialogue window, where the selected anchor can be modified
- Remove — clicking the left mouse button on an anchor opens the dialogue window to confirm the reinforcement removal – confirming this action then removes the anchor

Effect of anchors on the analysis is described in more details in the theoretical part of the hint.

Note: The program does not check the anchor bearing capacity against breakage.



Frame "Anchors"

Reinforcements

The "Reinforcement" frame contains a table with a list of inputted reinforcements. Adding (editing) reinforcement is performed in the "New reinforcement (Modify interface properties)" dialogoue window. The inputted reinforcements can be edited on the desktop with the help of active objects.

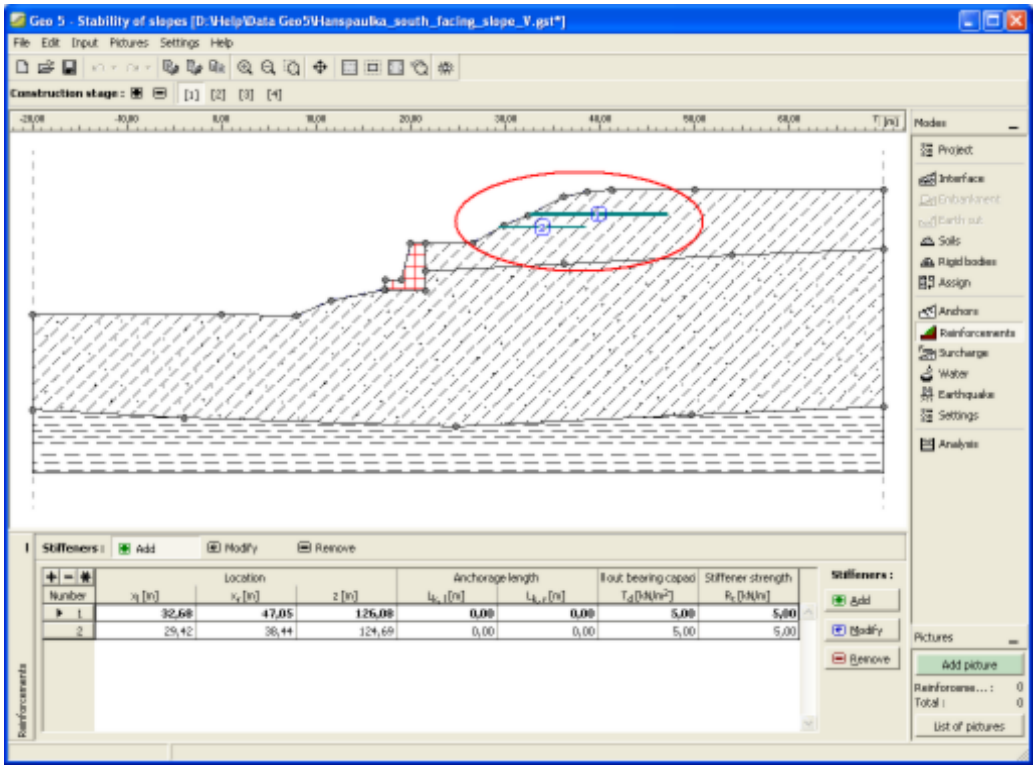
Properties like reinforcement location, anchorage length from both left and right end and reinforcement strength must be specified. All inputted parameters can be modified only in the stage of construction, in which the reinforcement was introduced. In subsequent stages the geo-reinforcement can only be removed.

Reinforcements can also be introduced with help of a mouse click. Mouse input mode is determined by pressing buttons on the horizontal bar "Reinforcement". The following modes are available:

- Add — clicking the left mouse button allows for specifying the starting and end point of a reinforcement. A predefined grid can be used in the input mode. The starting point is always "attached" to the terrain. The coordinates of inputted points are automatically round up to two digits – both input modes (manual, mouse) are therefore identical
- Modify — clicking the left mouse button on already existing reinforcement opens the "Modify reinforcement property" dialogue window, where the selected reinforcement can be modified
- Remove — clicking the left mouse button on reinforcement opens the dialogue window to confirm the reinforcement removal – confirming this action then removes the reinforcement

Reinforcements can be introduced only in the horizontal direction.

Including reinforcements in the analysis is described in more details in the theoretical part of the hint.



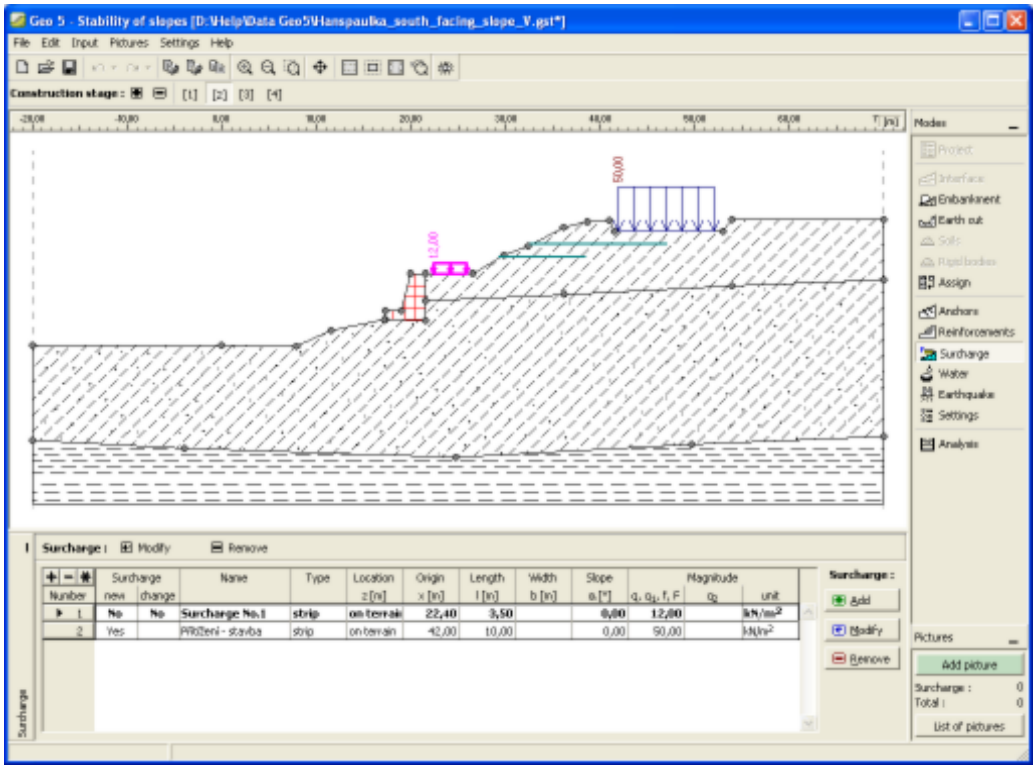
Frame "Reinforcements"

Surcharge

The "Surcharge" frame contains a table with a list of inputted surcharges. Adding (editing) surcharge is performed in the "New (edit) surcharge" dialogue window. The inputted surchages can be edited on the desktop with the help of active dimensions or active objects, respectively.

All inputted parameters of a surcharge can be modified in the construction stage where the surcharge was specified. Only the surcharge magnitude can be modified in all subsequent construction stages (option "Adjust surcharge").

Influence of surcharge on stability analysis of slopes is described in the theoretical part of the hint.



Frame "Surcharge"

Water

The "Water" frame serves to set the type of ground water table. Five options to specify the type of water are available from the combo list.

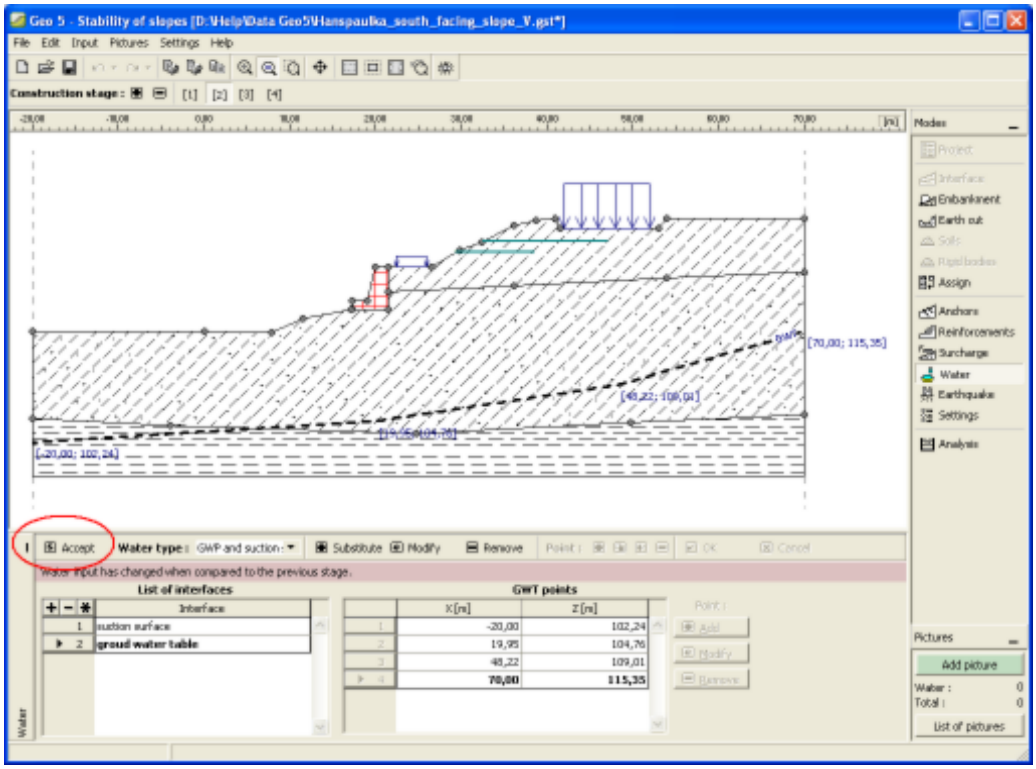
Inputting the ground water table or isolines, respectively, is identical with the standard input of interfaces.

A field for specifying a value of coefficient Ru or pore pressure appears next to the table if introducing water using isolines of Ru-interfaces or pore pressure, respectively. Pressing the button with a blue arrow next to the input field opens the "Coefficient Ru" or "Pore pressure" dialogue window to enter the desired value. It is advantageous to input all values at once using the "OK+↑" and "OK+↓". The value of a given quantity found in a specific point between two isolines is approximated by linear interpolation of values pertinent to given isolines. The first (the most top one) always coincides with terrain – it therefore cannot be deleted.

The ground water table or table of suction, respectively, is specified as continuous interfaces, which can be located even above the terrain.

If the inputted data in individual stages are different, the program then allows for accepting the data from the previous stage of construction by pressing the " Accept" button.

The program further allows for specifying a depth of tensile cracks filled with water.

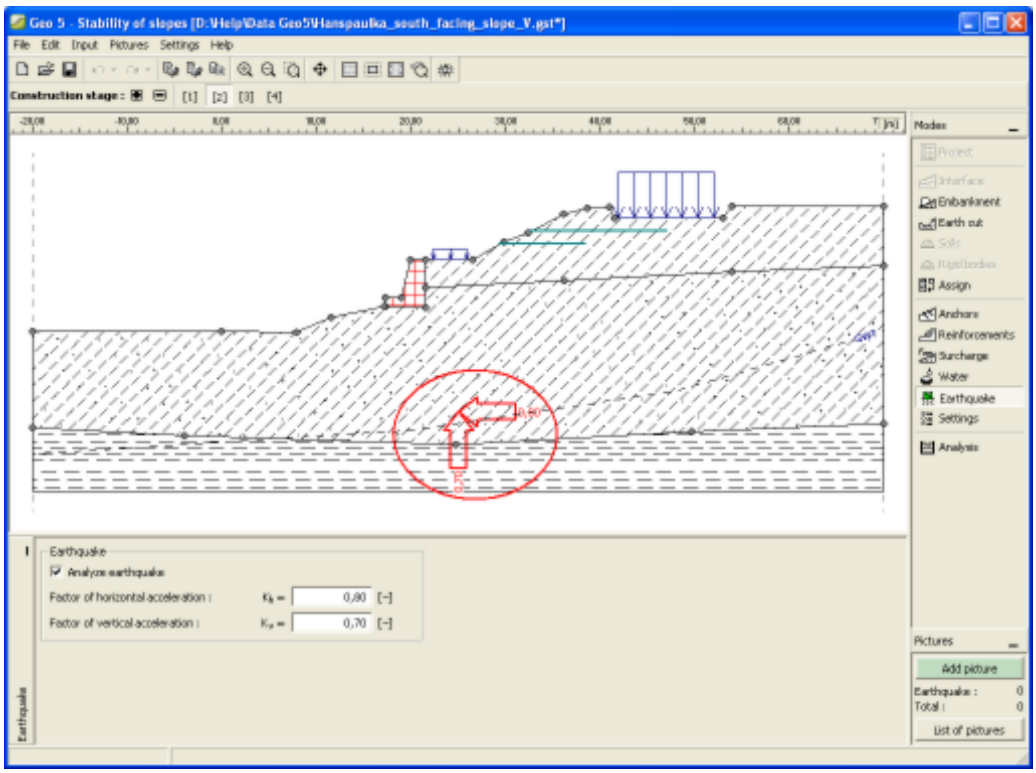


Frame "Water"

Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of inputted earthquake effects are displayed on the desktop.

Slope stability analysis while accounting for earthquake is described in the theoretical part of the hint in chapter "Influence earthquake".



Frame "Earthquake"

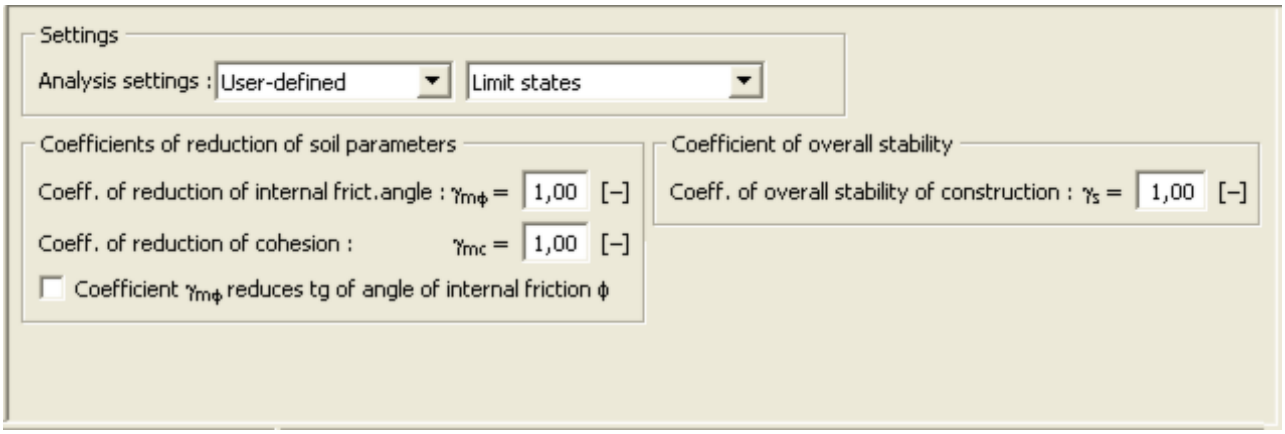
Setting

The "Settings" frame contains basic settings to assess the slope stability. The program offers pre-setting for different countries (Czech Republic, Germany, France,...) and "Standard setting" recommended by the authors of the program for countries no included in the list. While changing settings in the combo list the values of coefficients of reduction of soil parameters in corresponding windows are changed. Changing settings in the combo list modifies previously set values in corresponding windows.

An arbitrary analysis setting is available with the option "User setting".

Selecting the option "Safety factor" allows the user to specify in input field own value of the factor of safety.

Selecting the option "Limit states" allows for specifying individual values of the coefficients of reduction of soil parameters (e.g., recommended values of coefficients according to EC7-1) and coefficient of overall stability.


Settings
Analysis settings : User-defined
Limit states
Coefficients of reduction of soil parameters
Coeff. of reduction of internal frict.angle : γmφ = 1,00 [–]
Coeff. of reduction of cohesion : γmc = 1,00 [–]
Coefficient γmφ reduces tg of angle of internal friction φ
Coefficient of overall stability
Coeff. of overall stability of construction : γs = 1,00 [–]

Frame "Settings"

Analysis

The "Analysis" frame displays the analysis results. Several analyses can be performed for a single task.

The starting point in the slope stability analysis is the selection of the type of slip surface. The input is available from a combo list in the left top part of the frame containing two options – circular slip surface and polygonal slip surface. After introducing the slip surface the analysis is started using the "Analysis" button. The analysis results appear in the right part of the frame.

The type of analysis is selected in the mid section of the frame – the circular slip surface allows two options – the Bishop or Petterson method, the polygonal slip surface is analyzed exploiting the Sarma method.

Based on the "Settig" in the frame the verification of the slope stability is performed using either the factor of safety concept or the theory of limit states.

Checking the box "Optimize slip surface" allows for optimizing either the circular or polygonal slip surface. This step also activates the "Constrains" button – pressing this button changes the frame appearance and makes possible to introduce constrains on optimization procedure.

It is also possible to specify how to deal with anchors in the analysis (box "Infinite anchor lengths assumed").

The slip surface (even the optimized one) must be introduced in the frame – several possibilities are available:

- Circular slip surface

using mouse – press the "Input" button to active the input regime and then by clicking the left mouse button enter three points to define a circular slip surface (the introduced slip surface can be further modified using the "Modify" button or specified again with the help of the "Replace" button

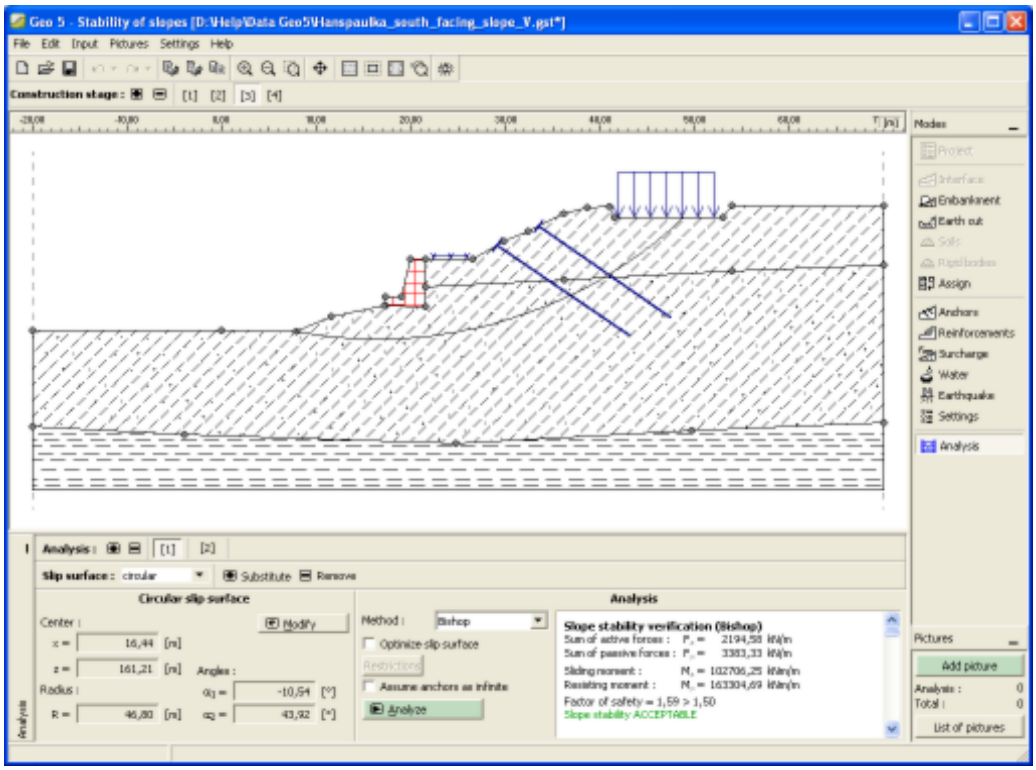
using the dialogue window – pressing the "Input" button in the "Circular slip surface" frame opens the "Circular slip surface" dialogue window that allows for specifying the radius and coordinates of center

- Polygonal slip surface

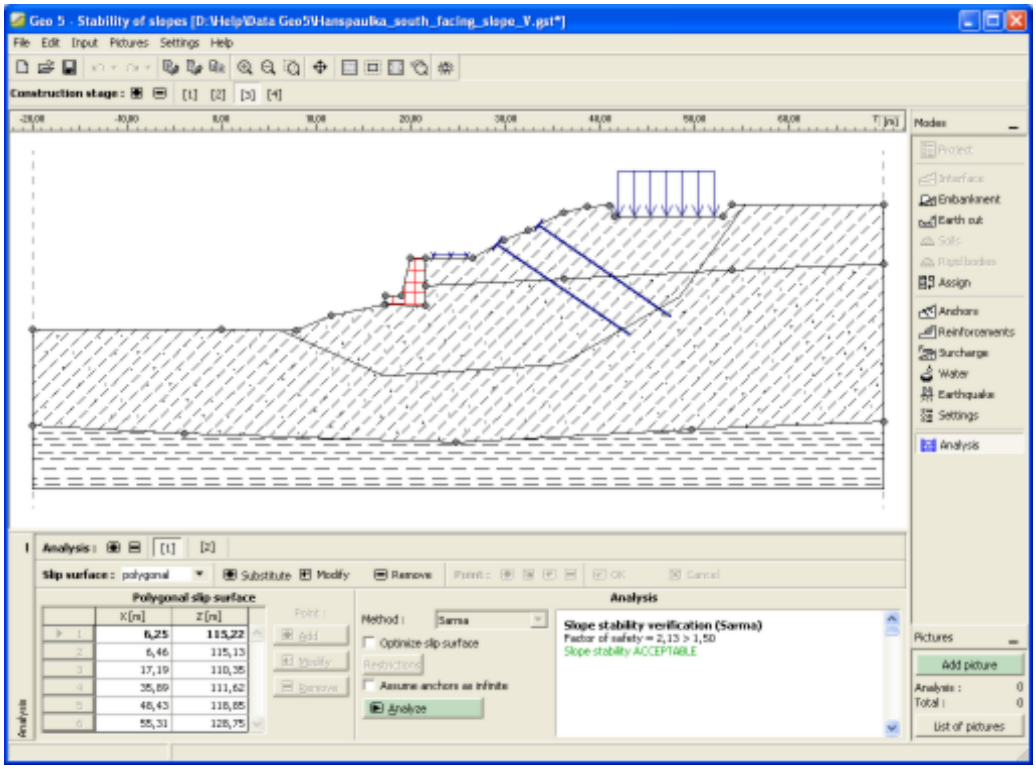
  using mouse - press the "Input" button to active the input regime – entering the surface points proceeds in the same steps as when specifying interfaces

  using table – pressing the button "Input" actives several buttons that allow for filling the adjacent table with the coordinates of slip surface points (buttons "Add", "Modify", "Remove")

The analysis results appear in the left part of the frame and the optimized slip surface on the desktop. Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Analysis" - circular slip surface

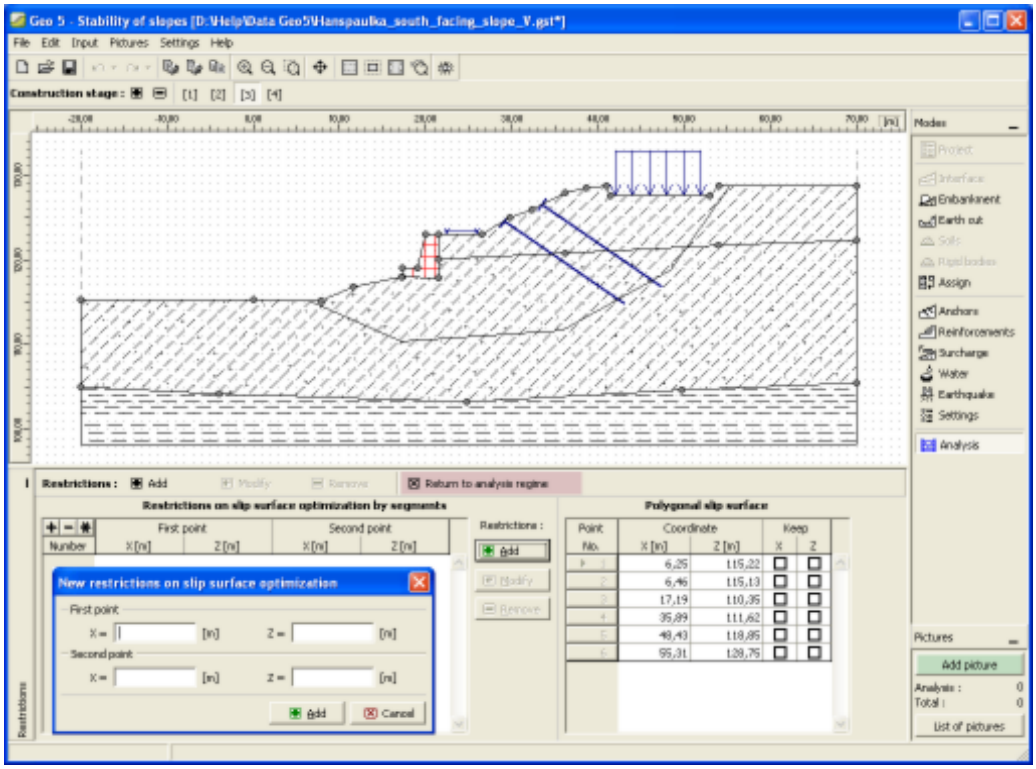


Frame "Analysis" - polygonal slip surface

Constrains on the optimization procedure

The "Analysis" frame allows (after pressing the "Constrains") for specifying constrains on the optimization process.
Regardless of the assumed type of slip surface it is possible to introduce into the soil body (with the help of mouse) segments, which should not be crossed by the optimized slip surface. These segments also appear in the table in the left part of the frame.

Polygonal slip surface also allows for excluding some points from optimization, either entirely or partially only in specified direction. "Keeping the point fixed" during optimization process is achieved by checking the box in the table with corresponding point. This input mode is quitted by pressing the red button "Return to analysis regime".

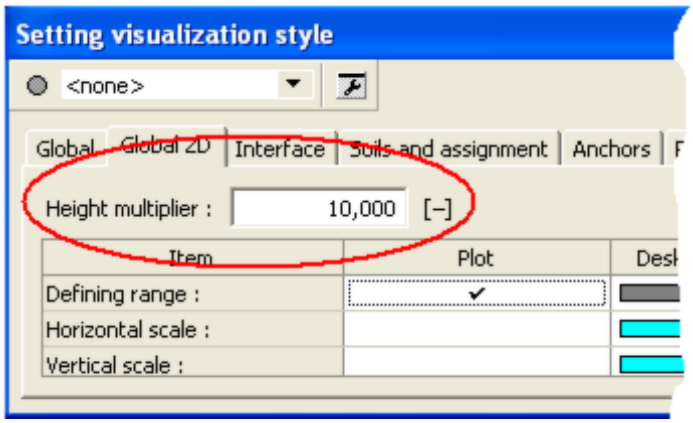


Frame "Analysis" - constrains on slip surface optimization by segments

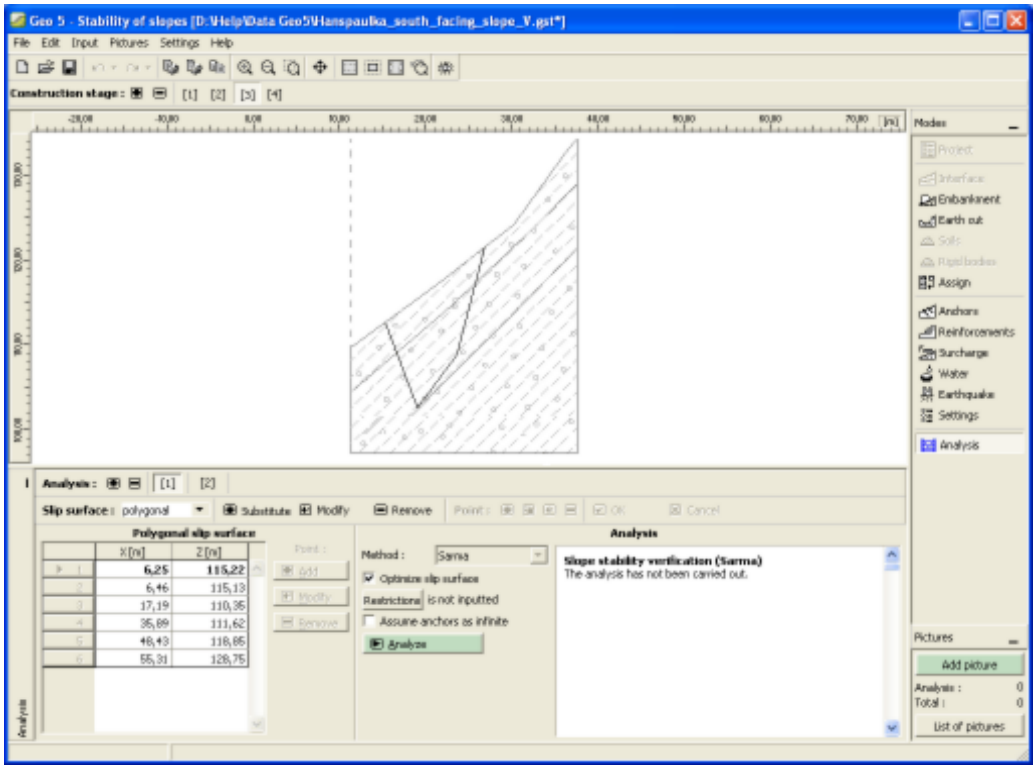
Height multiplier

Providing the analyzed slope is too long or has small height the plotted slip surface might not be sufficiently visible. This problem can be solved by selected courser scale in the vertical direction with the help of height multiplier. The value of this multiplier is set in the "Setting visualization style" dialogue window, tab sheet "Global 2D". Using standard setting ("Height multiplier" equal to one) plots undistorted structure proportional to its dimensions.

Only polygonal slip surface can be inputted graphically when exploiting the height multiplier option. The circular slip surface must be in such a case inputted manually in the "Circular slip surface" dialogue window using the "Input" button.



Setting height multiplier



Visualization of the resulting slip surface when using height multiplier

Program Cantilever Wall

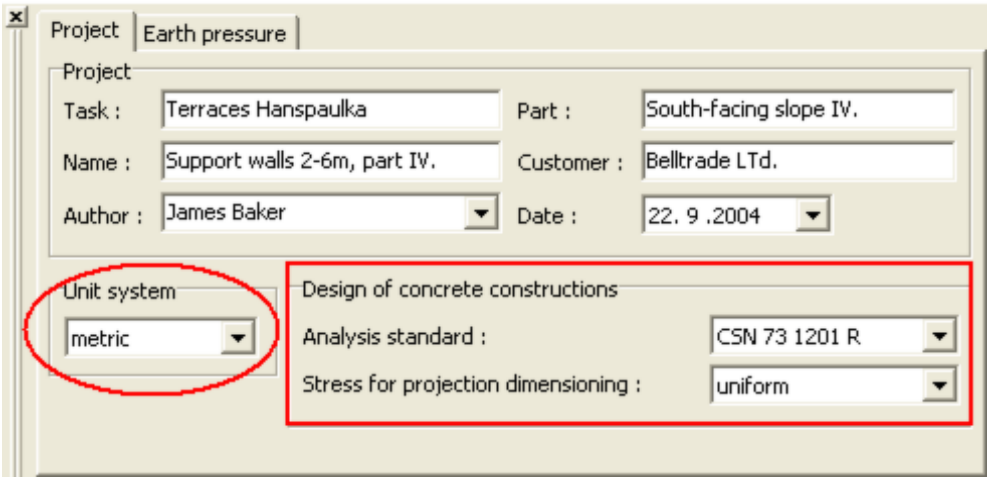
Project

The "Project" frame is used to input the basic project data and to specify the overall setting of the analysis run. The "Project" tab sheet contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc.

The "Project" tab sheet also allows the user to switch analysis units (metric / imperial).

The "Project" tab sheet further serves to specify a standard for concrete structures dimensioning. Referring to the selected standard the types of concrete and steel are then inputted in the "Material" frame. The dimensioning of cross-sections of the analyzed structure is then performed in the "Dimensioning" frame. Only ACI standard is available when selecting imperial units. While selecting standard it is also necessary to specify whether the stress in the foundation joint used for verification of a wall front key is assumed uniform (CSN) or trapezoidal (Eurocode).

The "Earth pressures" tab sheet serves to choose the basic theory or standards to be followed in the solution of a given problem.

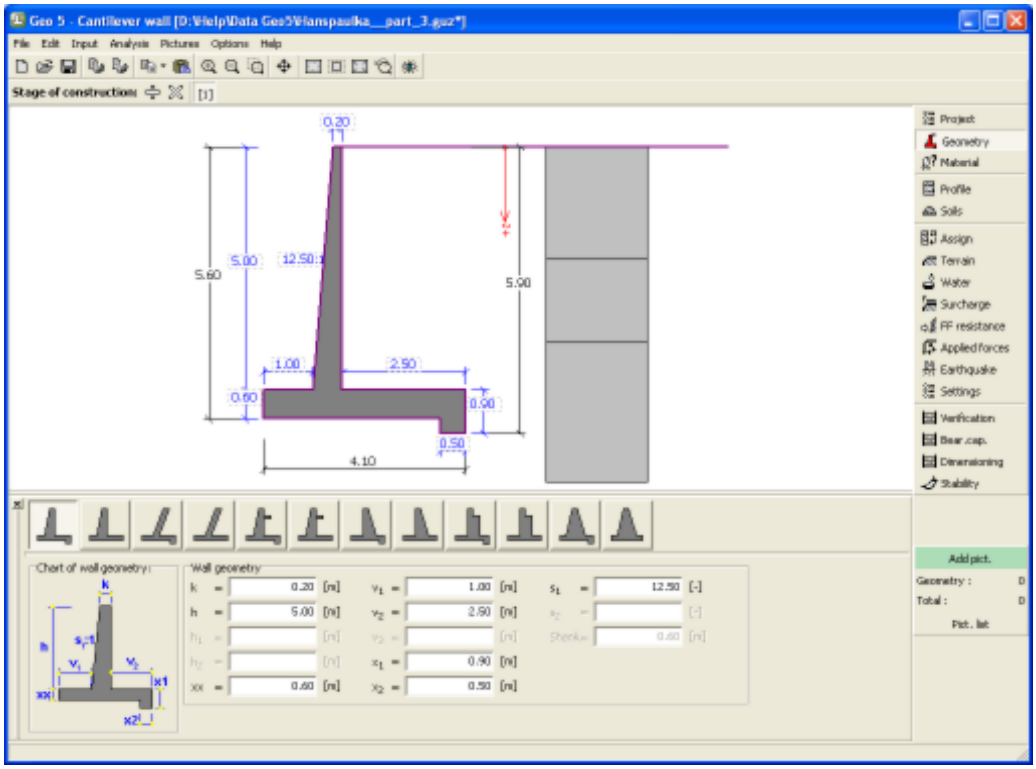


Frame "Project" - tab sheet "Project"

Geometry

The "Geometry" frame allows by pressing the button for selecting the wall shape. The selected shape with a graphic hint "Wall geometry chart" appears in the left part of the frame. The shape of a wall can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

In case the structure is composed of inclined segments it is required to enter the ratio of sides of an inclined segment 1:x. The straight structure is specified by entering the value zero.



Frame "Geometry"

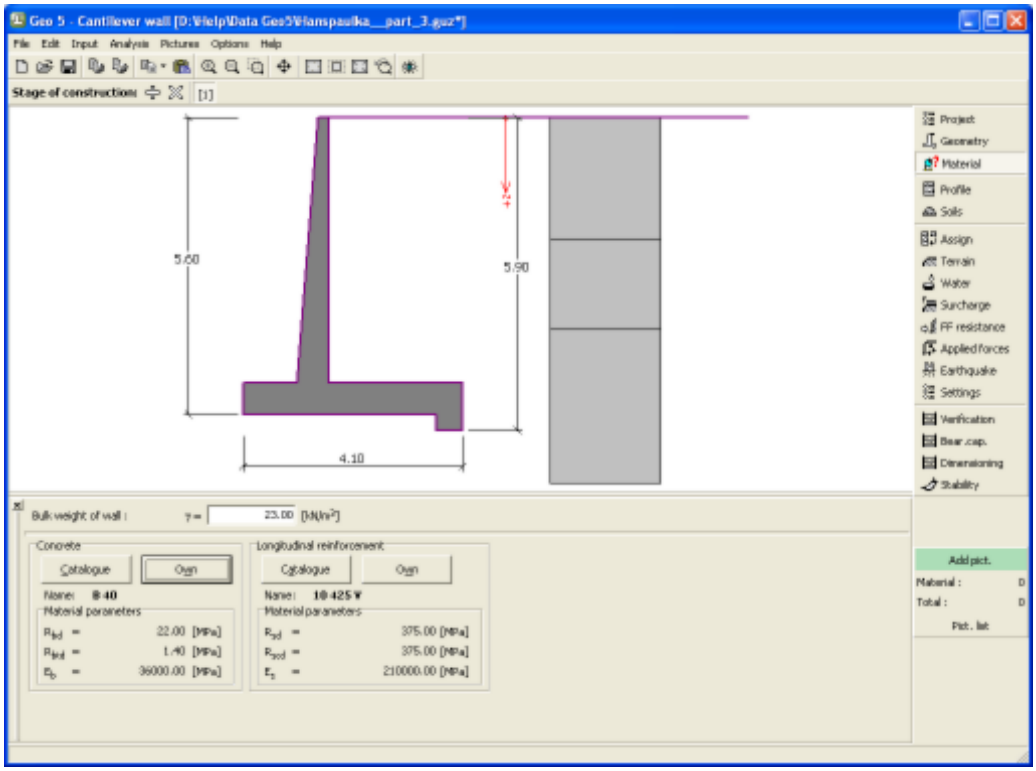
Material

The "Material" frame allows for the selection of material parameters for concrete and longitudinal steel reinforcements.

Two options are available when selecting the material type:

- the "Catalogue" button opens the "Material catalogue" dialogue window (for concrete or steel reinforcements), the list of materials then serves to select the desired material
- the "User" button opens the "Edit material – concrete" dialogue window (for concrete) or the "Edit material – concrete steel" dialogue window (for longitudinal steel reinforcements), which allows for manual specification of material parameters

The catalogues content depends on the selection of standard for the design of concrete structures set in the "Project" frame. The input field in the upper part of the frame serves to specify the wall unit weight.



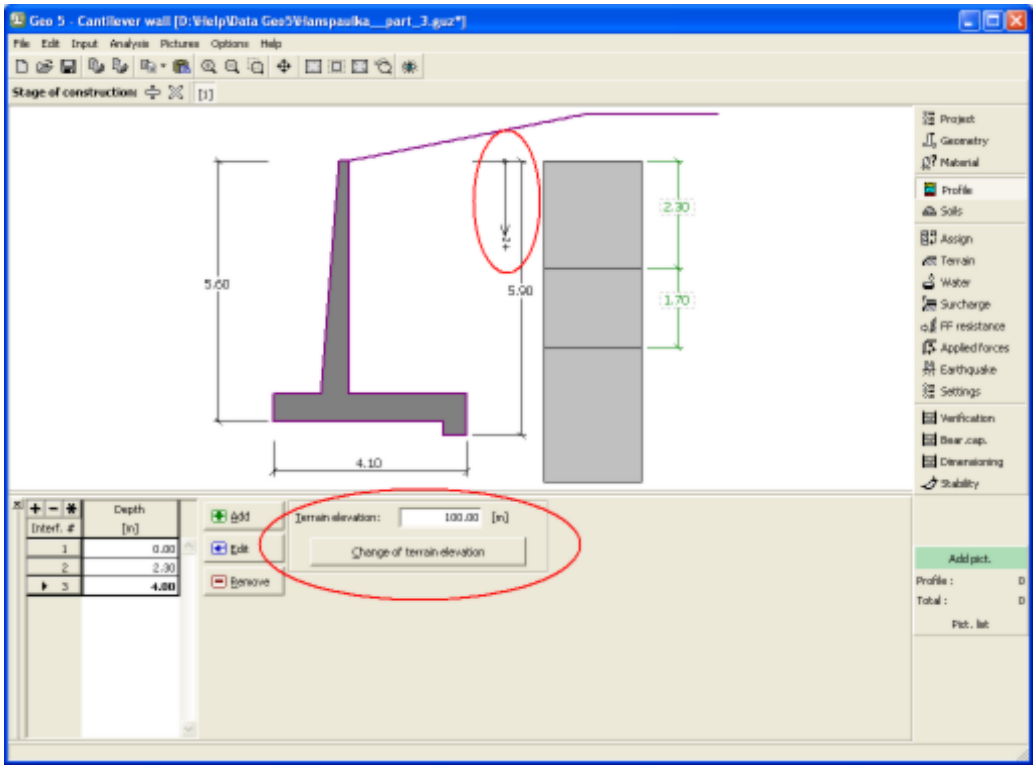
Frame "Material"

Profile

The "Profile" frame contains a table with a list of inputted interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "Add (edit) interface" dialogue window. The z-coordinate measured from the top point of a structure is specified.

The program allows for raising or lowering the top point of a structure in the "Change terrain elevation" dialogue window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".



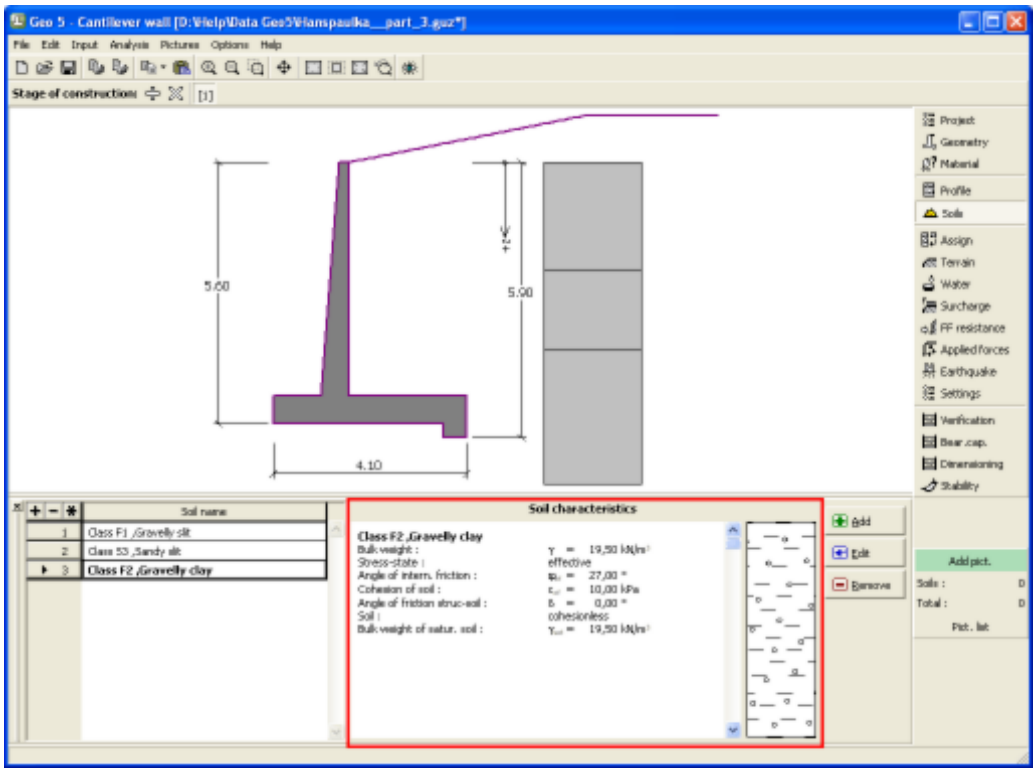
Frame "Profile"

Soils

The "Soils" frame contains a table with a list of inputted soils. The table also provides information about currently selected soil displayed in the right part of the frame.

Adding (editing) a soil is performed in the "Add (edit) soil" dialogue window.

The soil characteristics are specified in the program "Cantilever wall". These characteristics are further specified in chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

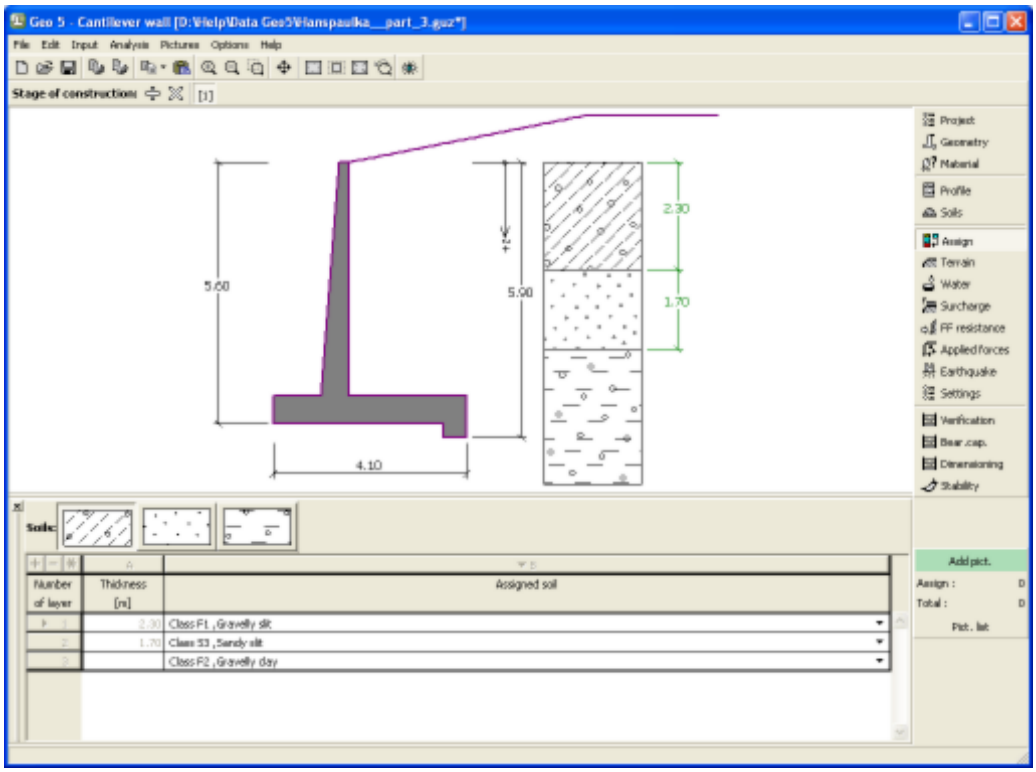


Frame "Soils"

Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

Procedure to assign soil into a layer is described in details herein.



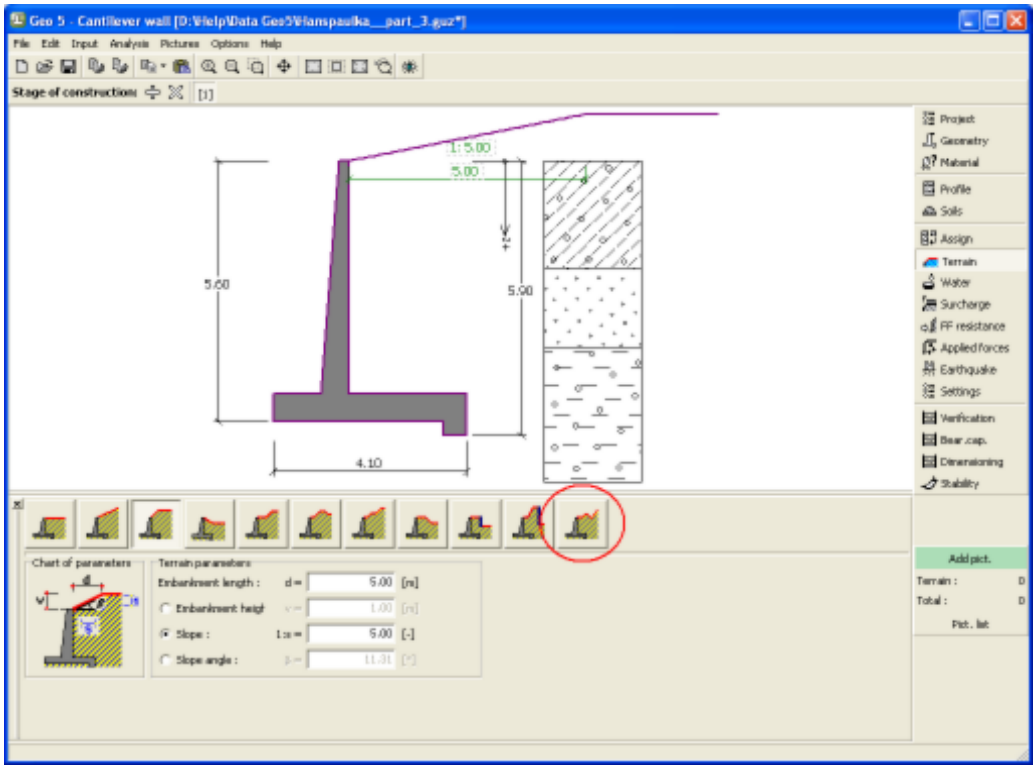
Frame "Assign"

Terrain

The "Terrain" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0, 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the hint, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

The "Water" allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. Water parameters (h1, h2...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

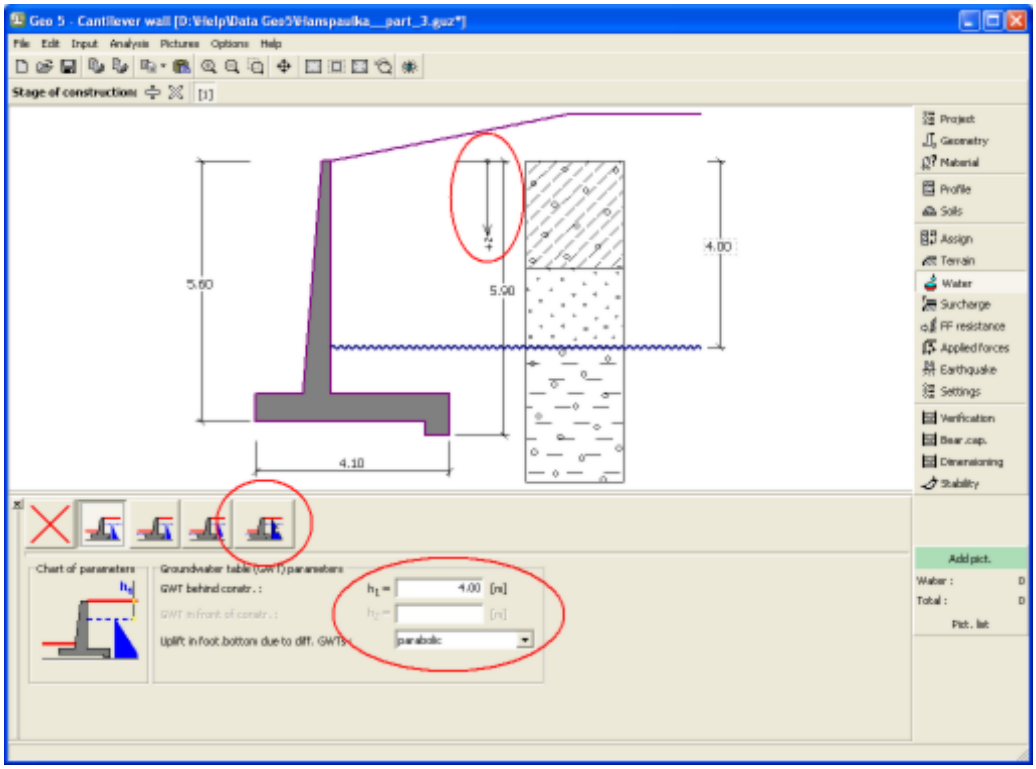
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic or it may not be considered at all. When verifying the wall, the uplift pressure in base of footing joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tab sheets "In front of structure" and "Behind structure" appear with tables. The table is filled with values of pore pressure in front, or behind the structure at a depth of "z" (z-axis).

The ground water table can also be specified above the structure or earth profile, respectively – in such a case the depth of water is inputted with negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the hint chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



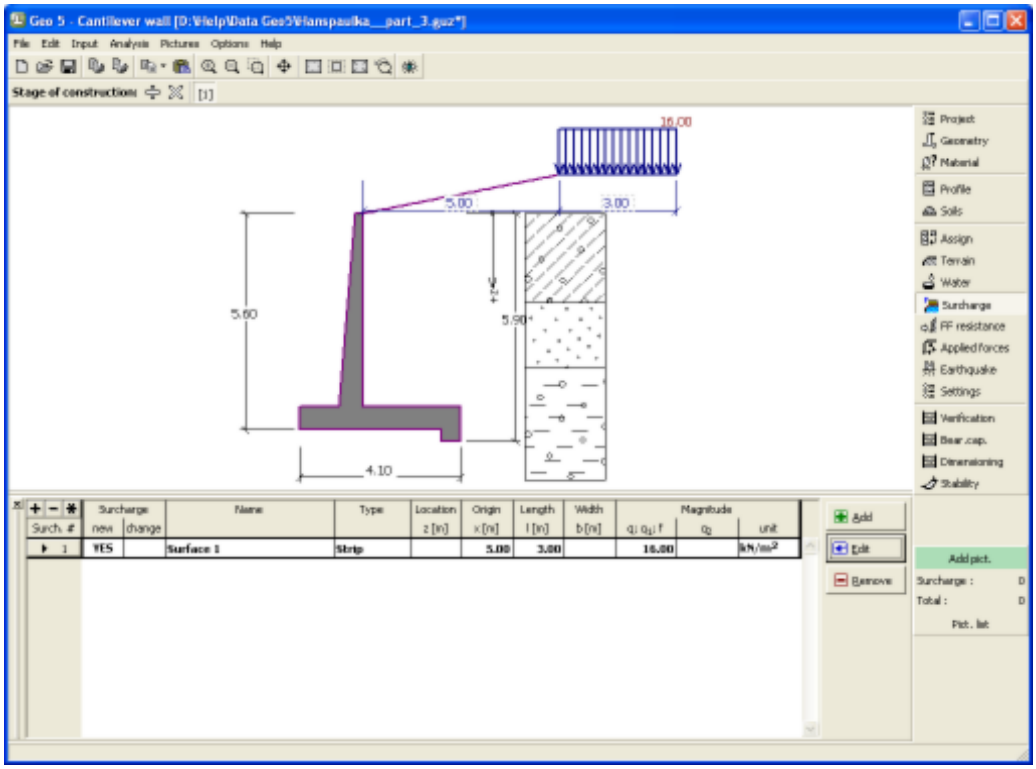
Frame "Water"

Surcharge

The "Surcharge" frame contains a table with a list of inputted surcharges. Adding (editing) surcharge is performed in the "New (edit) surcharge" dialogue window. The inputted surchages can be edited on the desktop with the help of active dimensions or active objects, respectively.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. Providing the surcharge is found off the terrain the computer prompts an error message.

Analysis of earth pressures due to surcharges is described in the theoretical part of the hint, chapter "Influence of surcharge".



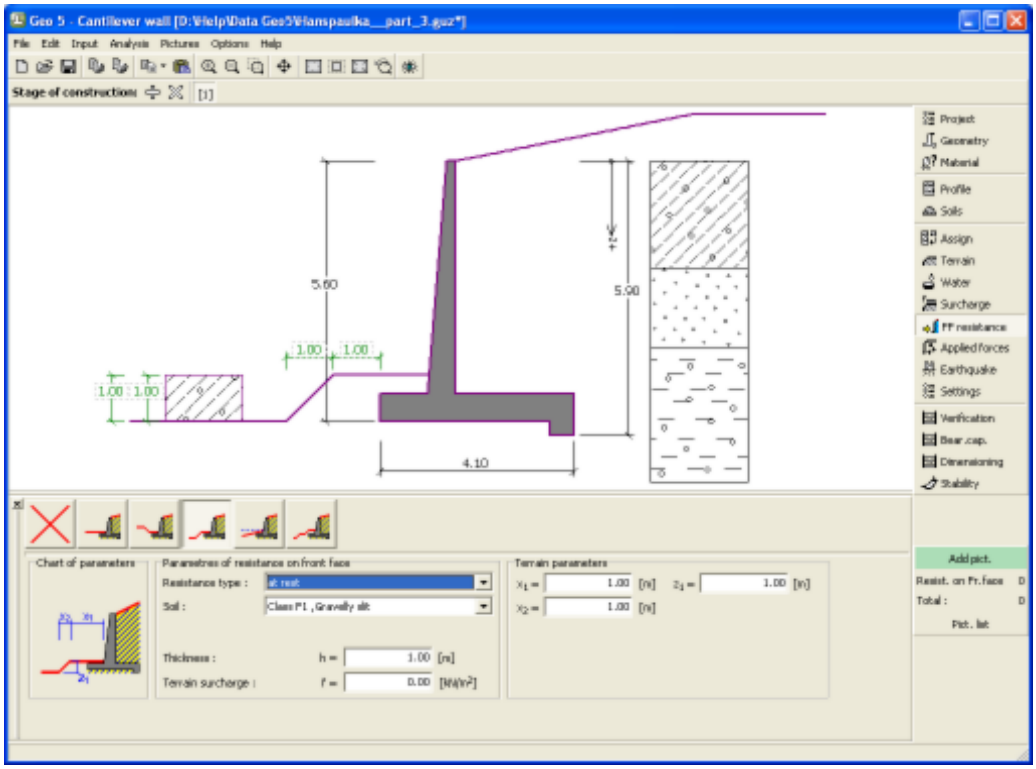
Frame "Surcharge"

Front face resistance

The "Front face resistance" frame allows by pressing the button for specifying the terrain shape and parameters of front face resistance. The selected shape with a graphic hint (" Parameter chart") of inputted values are displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Combo lists in the frame allows the user to select the type of resistance and a soil (the c ombo list contains soils introduced in the regime "Soils"). The magnitude of terrain surcharge in front of the wall or soil thickness above the wall lowest points can also be specified in the frame.

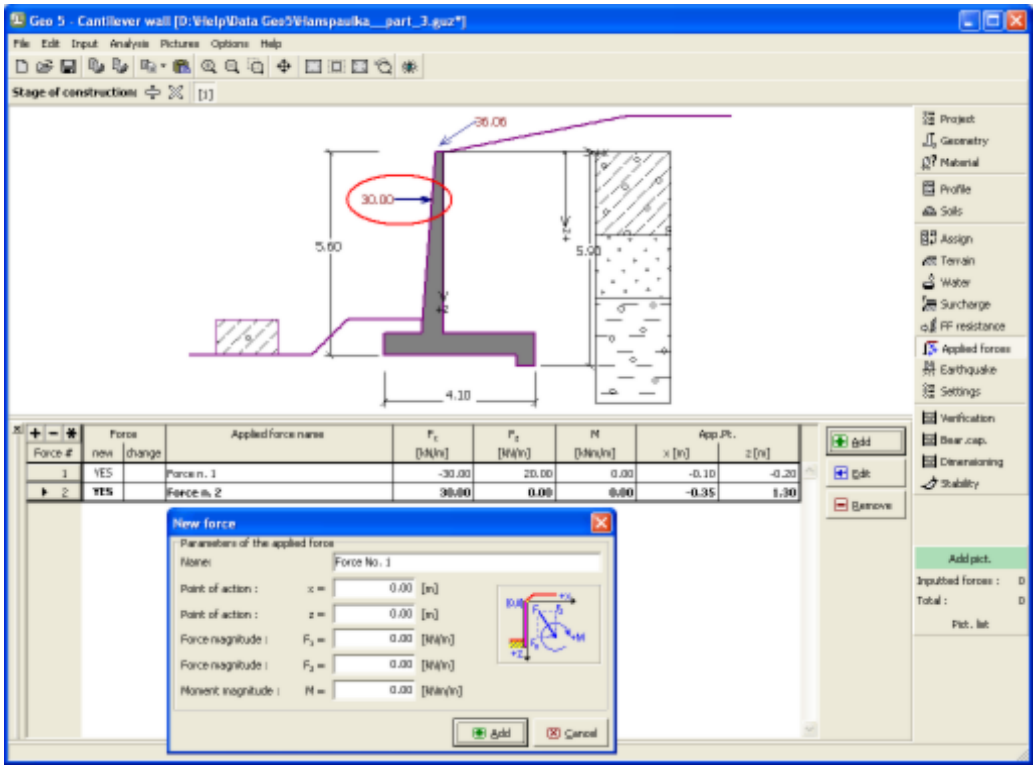
The resistance on a structure front face can be specified as a pressure at rest, passive pressure or reduced passive pressure. The resulting force due to reduced passive pressure is found as a resultant force caused by passive pressure multiplied by a corresponding coefficient, which follows from the inputted type of reduced passive pressure.



Frame "Front face resistance"

Inputted forces

The "Inputted forces" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "New force (edit force)" dialogue window. The inputted forces can also be edited on the desktop with the help of active objects.

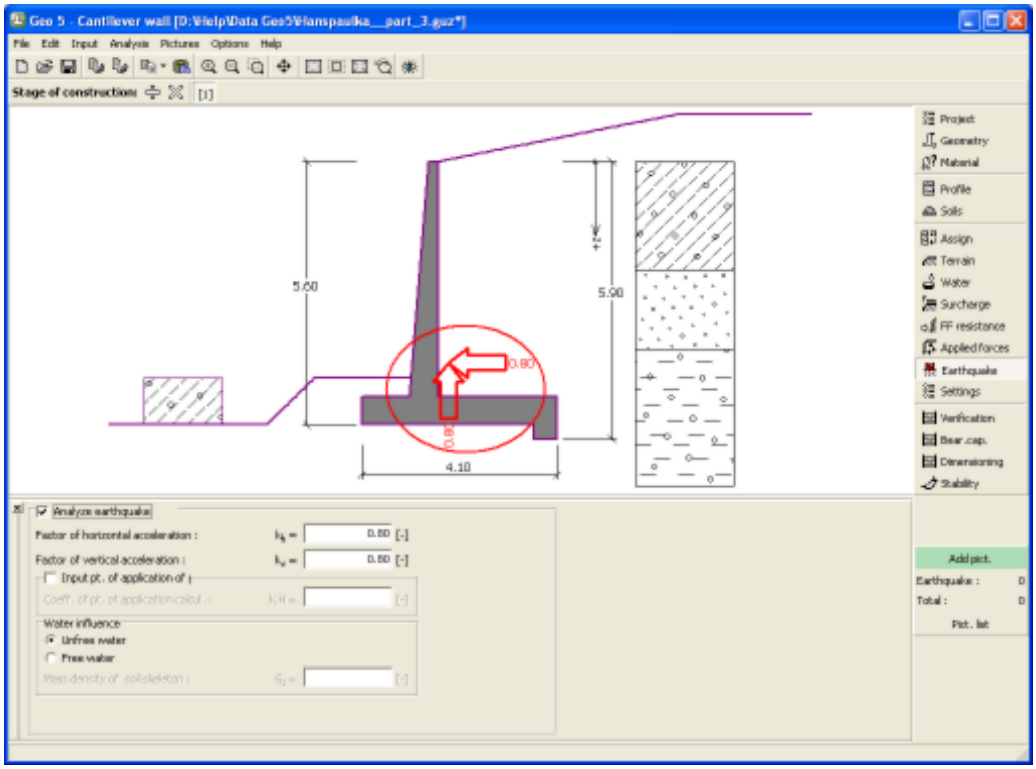


Frame "Inputted forces"

Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of inputted earthquake effects are displayed on the desktop.

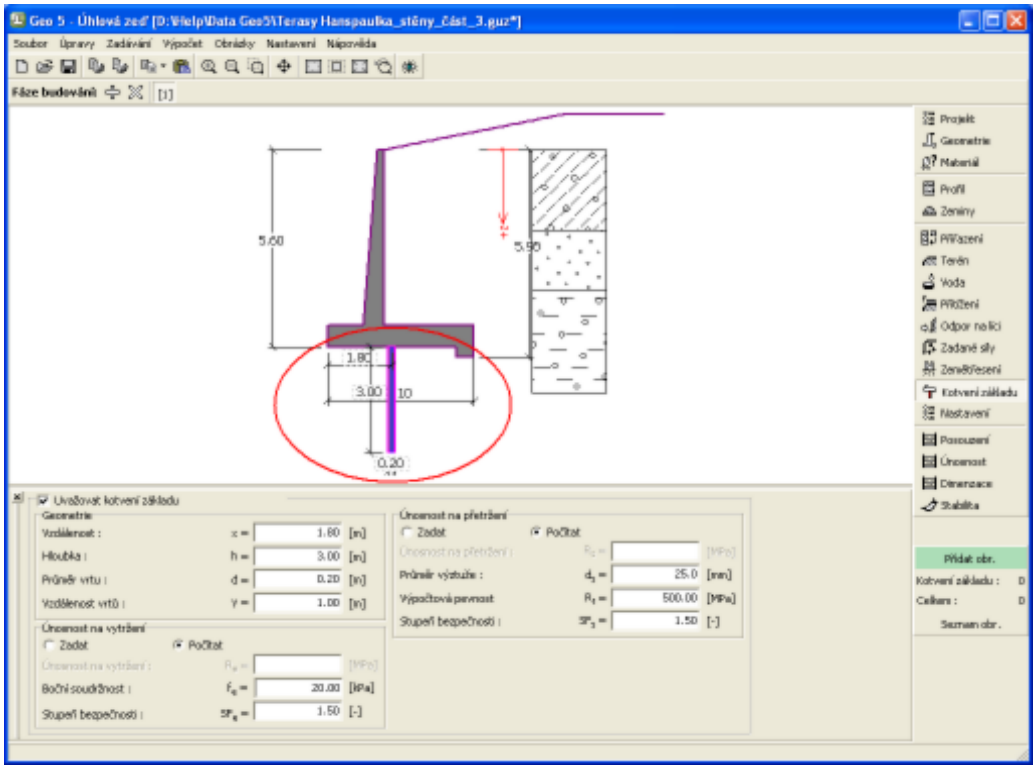
Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the hint in chapter "Influence earthquake".



Frame "Earthquake"

Base anchorage

The frame "Base anchorage" serves to input parameters (anchorage geometry, bearing capacity against pulling-out and pulling-apart) specifying an anchorage of the wall foundation. Geometry of footing anchorage can be edited either in the frame by inserting values in the inputting boxes or on the desktop with the help of active dimensions. The bearing capacity values can be either inputted or computed by the program from the inputted parameters.



Frame "Base anchorage"

Setting

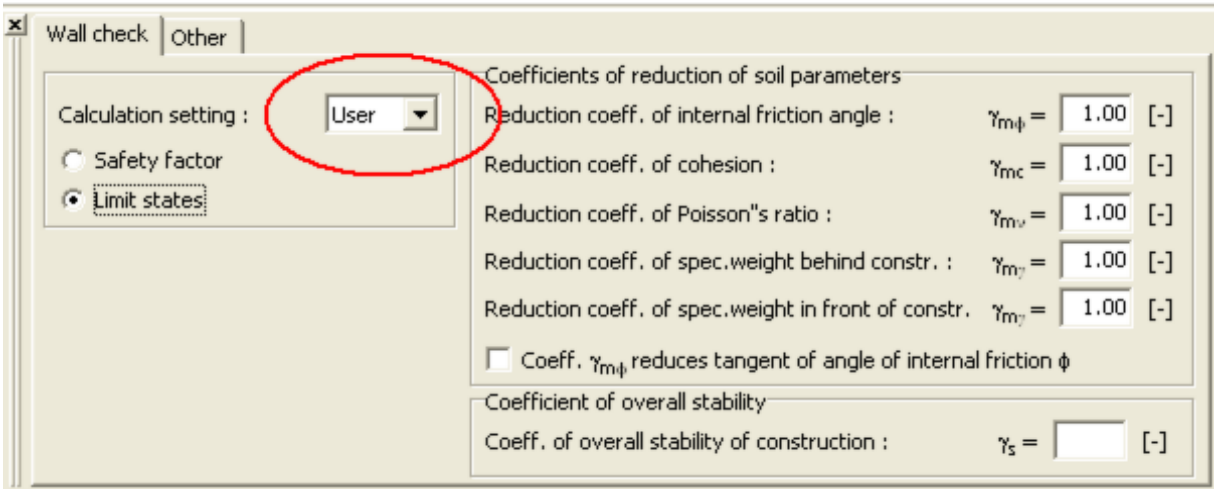
The "Setting" frame contains basic settings for the analysis of earth pressures. The program offers pre-setting for different countries (Czech Republic, Germany, France,...) and " Standard setting" recommended by the authors of the program for countries no included in the list. While changing settings in the combo list the values of coefficients of reduction of soil parameters in corresponding windows are changed.

Evaluating the structure according to theory of limit states also calls for the input of coefficient of overall stability of a structure. When subjecting the wall to overall verification this coefficient is used to multiply the resisting moment M_{res} and the resisting shear force H_{res}.

An arbitrary analysis setting is available with the option "User setting". Selecting the option "Reduce soil parameters" allows for specifying in input fields individual values of the coefficients of reduction of soil parameters (e.g., recommended values according to EC7-1).

The "Other" tab sheet serves to specify the type of pressure acting on a wall based on the allowable wall deformation. Providing the wall is free to move an active pressure is assumed, otherwise, a pressure at rest is used.

For wall with footing jump it is possible to choose in the tab sheet "Other" the way of accounting for the wall footing jump.


Wall check
Other
Calculation setting :
User
Safety factor
Limit states
Coefficients of reduction of soil parameters
Reduction coeff. of internal friction angle : γmφ = 1.00 [-]
Reduction coeff. of cohesion : γmc = 1.00 [-]
Reduction coeff. of Poisson"s ratio : γmν = 1.00 [-]
Reduction coeff. of spec.weight behind constr. : γmγ = 1.00 [-]
Reduction coeff. of spec.weight in front of constr. γmγ = 1.00 [-]
Coeff. γmφ reduces tangent of angle of internal friction φ
Coefficient of overall stability
Coeff. of overall stability of construction : γs = [-]

Frame "Setting"

Verification

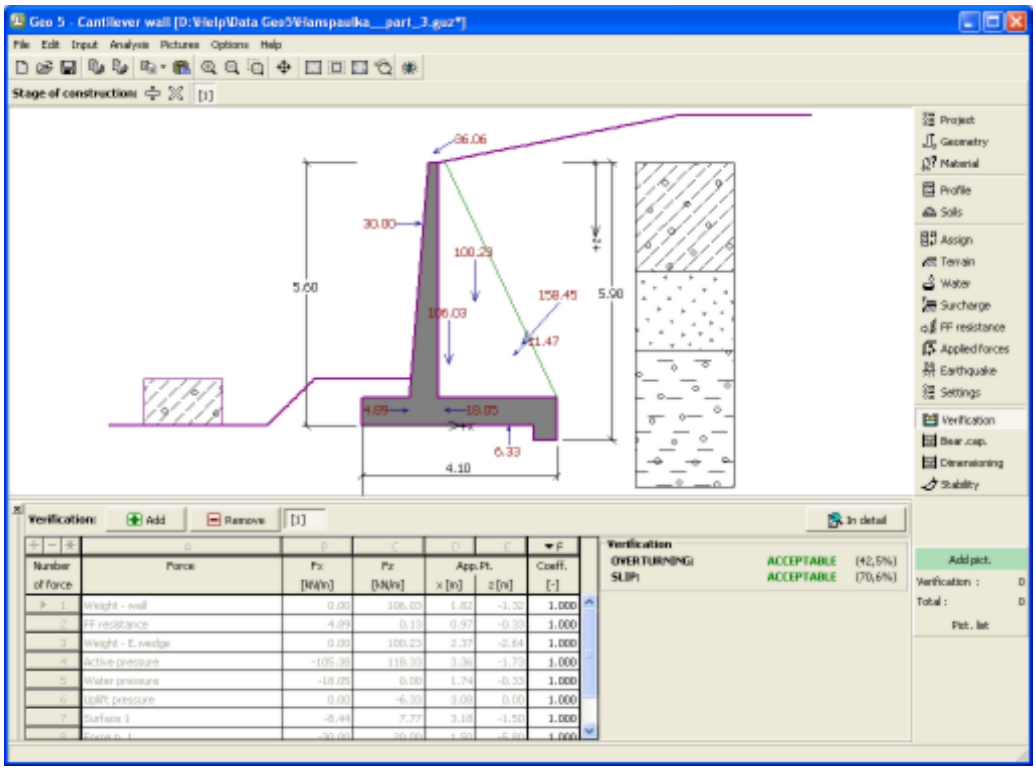
The "Verification" frame shows the analysis results. Several computation with different coefficients of resultant force effects can be carried out for a single task.

The wall is loaded either by active pressure or pressure at rest depending on input in the frame "Setting".

Procedure for wall verification is described in the theoretical part of the hint.

The computed forced are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against overturning and translation. The "In detail" button opens the dialogue window, which contains detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Verification"

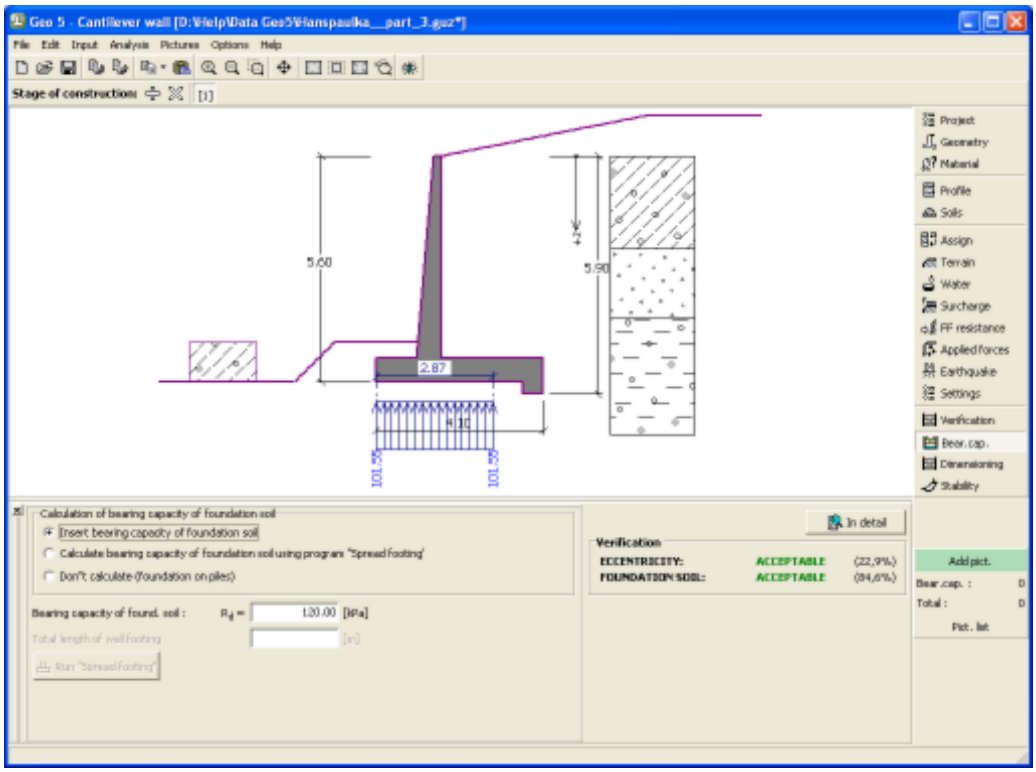
Bearing capacity

The "Bearing capacity" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "Verification". The program "Spread footing" then considers all verifications as loading cases.

Three basic analysis options are available in the frame:

- Input the foundation soil bearing capacity — The input field serves to specify the foundation soil bearing capacity. The results of verification analysis of a soil for eccentricity and bearing capacity are displayed in the right part of the frame. The "In detail" button opens the dialogue window that displays detailed listing of the results of verification analysis of foundation soil bearing capacity.
- Compute the foundation soil bearing capacity using the program "Spread footing" — Pressing the "Run "Spread footing" button starts the program "Spread footing" that allows for computing the soil bearing capacity or settlement and rotation of a footing. Pressing the "OK" button leaves the analysis regime – the results and all plots are copied to the program "Cantilever wall". The program "Spread footing" must be installed for the button to be active.
- Do not compute (pile footing) — The foundation soil bearing capacity is not computed.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Bearing capacity"

Dimensioning

The "Dimensioning" frame serves to design and verify the reinforcement of wall cross-section – the cross-section subjected to dimensioning is selected in the combo list.

- Wall stem verification

- Construction joint verification — depth of construction joint from construction top edge is specified
- Wall jump verification — type of assumed stress acting in construction joint for verification (linear, constant) can be selected in the frame "Project"

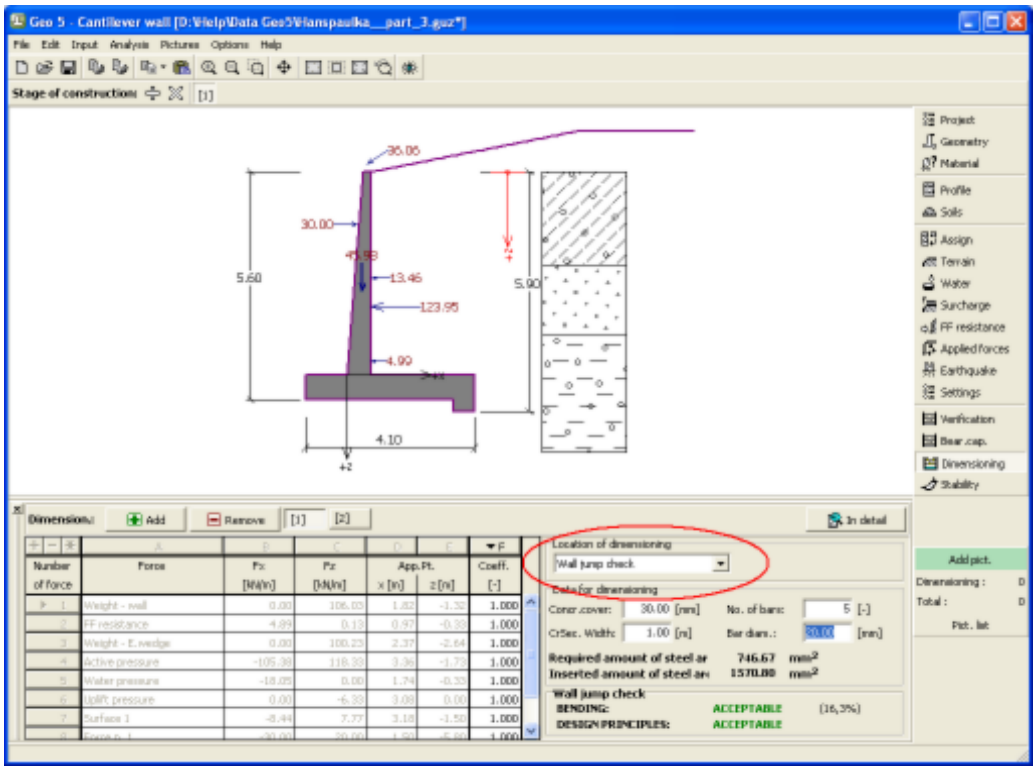
The wall stem and construction joint are always loaded by the pressure at rest. When verifying the front wall jump the wall is loaded either by the active pressure or the pressure at rest depending on input specified in the frame "Setting".

Procedure to derive distribution of internal forces in individual cross-sections is described in the theoretical part of this hint.

Dimensioning of the steel-reinforced concrete structure is performed according to the standard set in the frame "Project".

Several computations for various cross-sections can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "In detail" button opens the dialogue window that contains detailed listing of the dimensioning results.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.

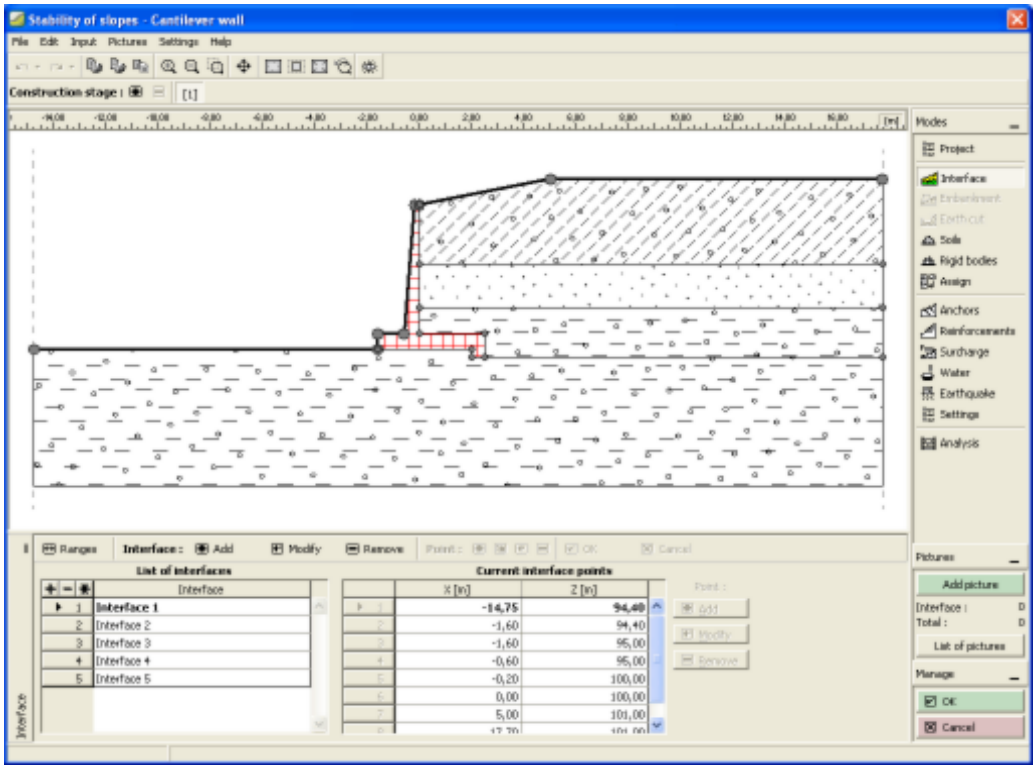


Frame "Dimensioning"

Stability

Pressing the "Stability" button launches the "Slope stability" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "Slope stability" is installed.

After completing all analyses press the "OK" button to leave the program – all data are then carried over to the analysis protocol of the "Cantilever wall" program.



Frame "Stability"

Program Masonry wall

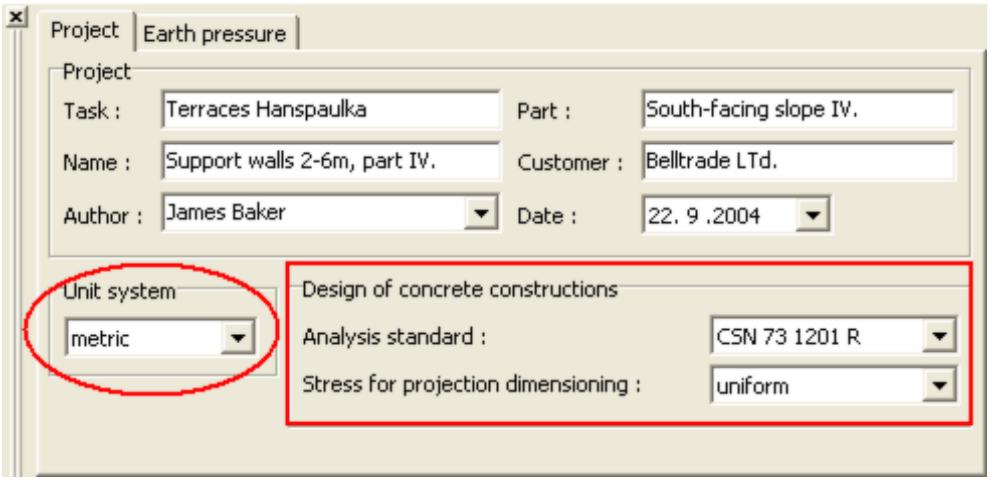
Project

The "Project" frame is used to input the basic project data and to specify the overall setting of the analysis run. The "Project" tab sheet contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc.

The "Project" tab sheet also allows the user to switch analysis units (metric / imperial).

The "Project" tab sheet further serves to specify a standard for concrete structures dimensioning. Referring to the selected standard the types of concrete and steel are then inputted in the "Materiál" frame. The dimensioning of cross-sections of the analyzed structure is then performed in the "Dimensioning" frame. Only ACI standard is available when selecting imperial units. While selecting standard it is also necessary to specify whether the stress in the foundation joint used for verification of a wall front key is assumed uniform (CSN) or trapezoidal (Eurocode).

The "Earth pressures" tab sheet serves to choose the basic theory or standards to be followed in the solution of a given problem.

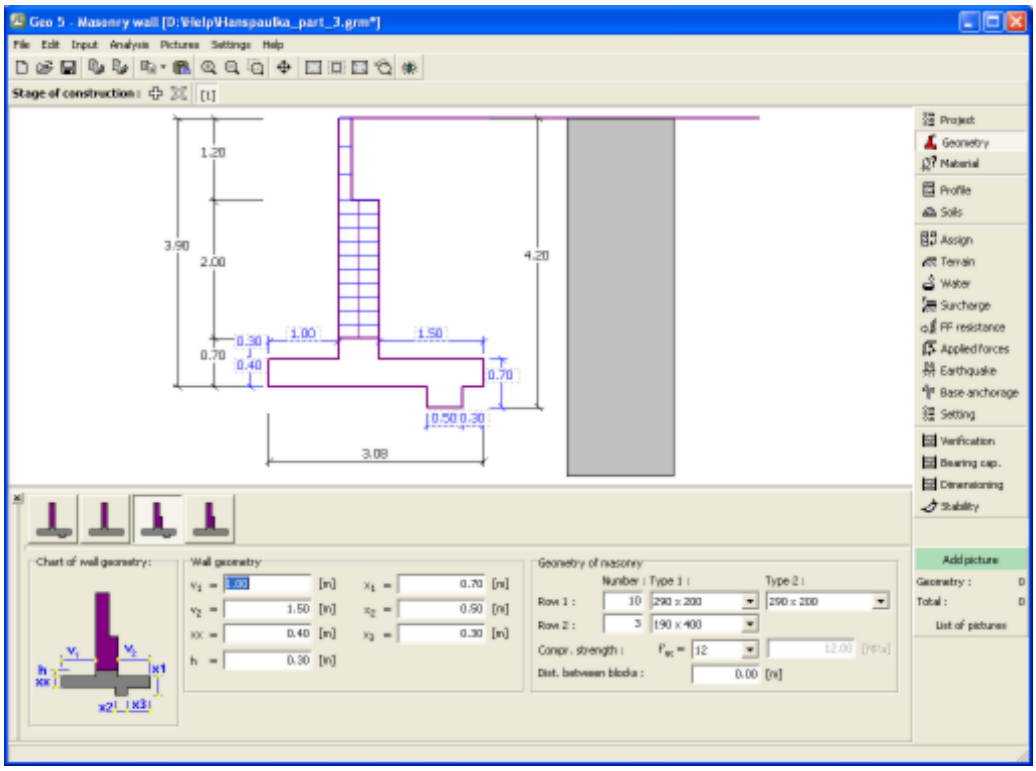


Frame "Project" - tab sheet "Project"

Geometry

The "Geometry" frame allows by pressing the button for selecting the wall shape. The selected shape with a graphic hint "Wall geometry chart" appears in the left part of the frame. The shape of a wall can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Based on the selected shape of a wall, you specify in the frame "Geometry of masonry" the number and dimensions of masonry blocks in individual columns, or if applicable also the thickness of vertical joint between blocks. In addition it is necessary to input compressive strength of masonry, which serves as the basic input parameter for the bearing capacity verification of reinforced masonry.



Frame "Geometry"

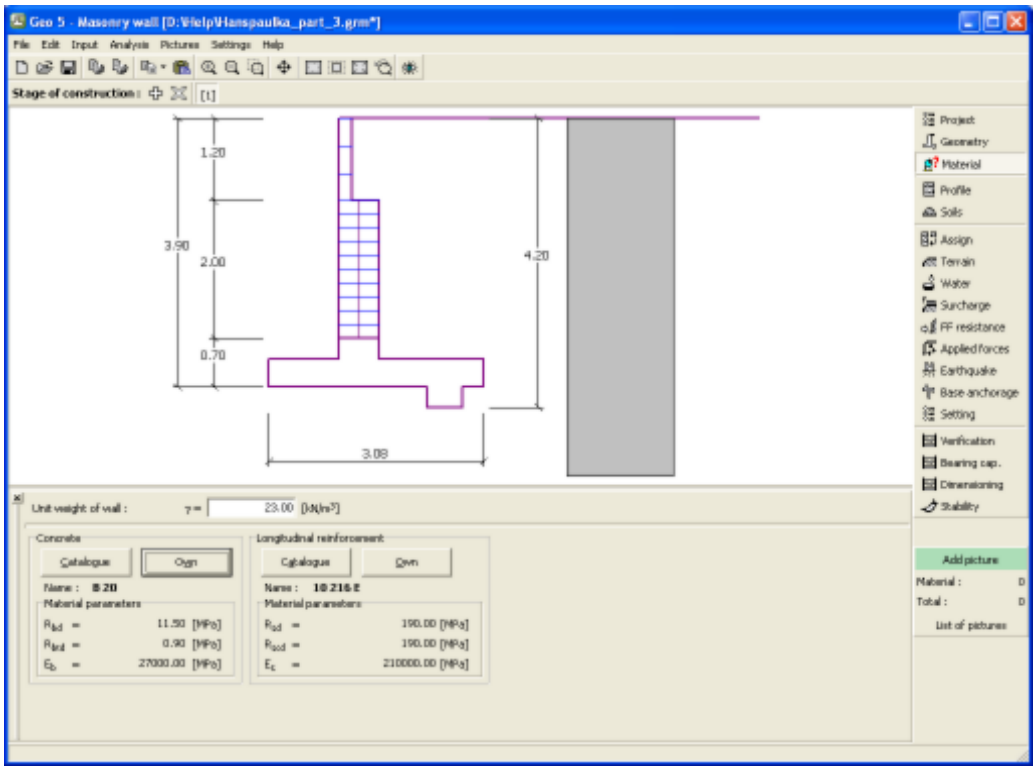
Material

The "Material" frame allows for the selection of material parameters for concrete and longitudinal steel reinforcements.

Two options are available when selecting the material type:

- the "Catalogue" button opens the "Material catalogue" dialogue window (for concrete or steel reinforcements), the list of materials then serves to select the desired material
- the "User" button opens the "Edit material – concrete" dialogue window (for concrete) or the "Edit material – concrete steel" dialogue window (for longitudinal steel reinforcements), which allows for manual specification of material parameters

The catalogues content depends on the selection of standard for the design of concrete structures set in the "Project" frame. The input field in the upper part of the frame serves to specify the wall unit weight.



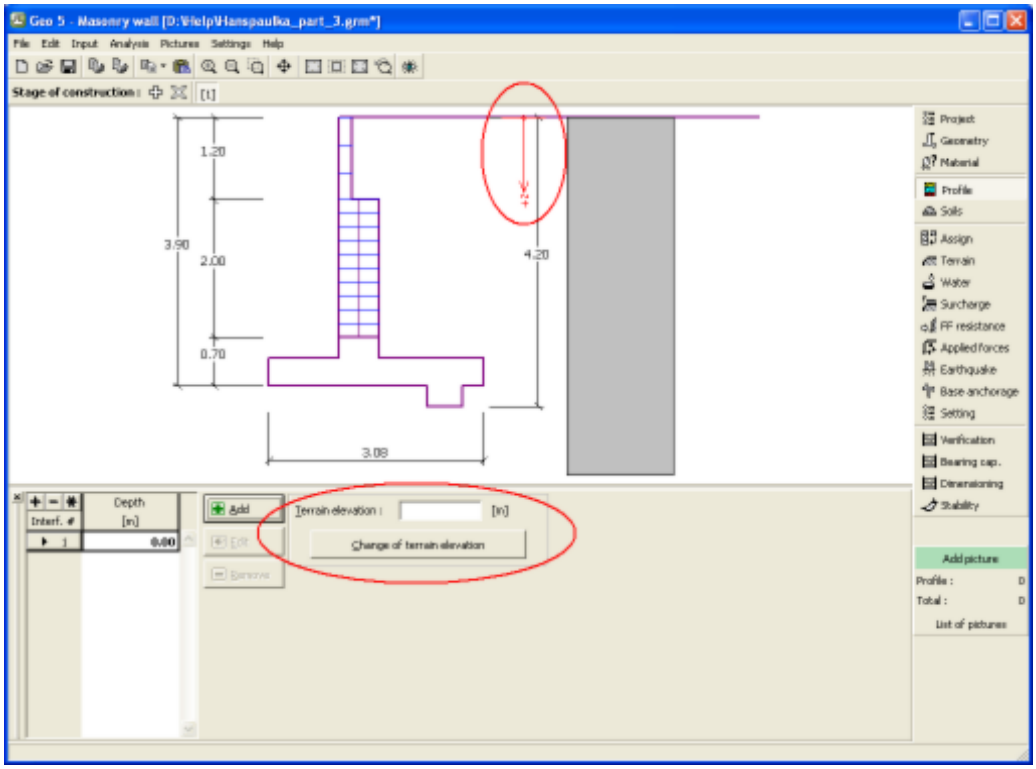
Frame "Material"

Profile

The "Profile" frame contains a table with a list of inputted interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "Add (edit) interface" dialogue window. The z-coordinate measured from the top point of a structure is specified.

The program allows for raising or lowering the top point of a structure in the "Change terrain elevation" dialogue window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".



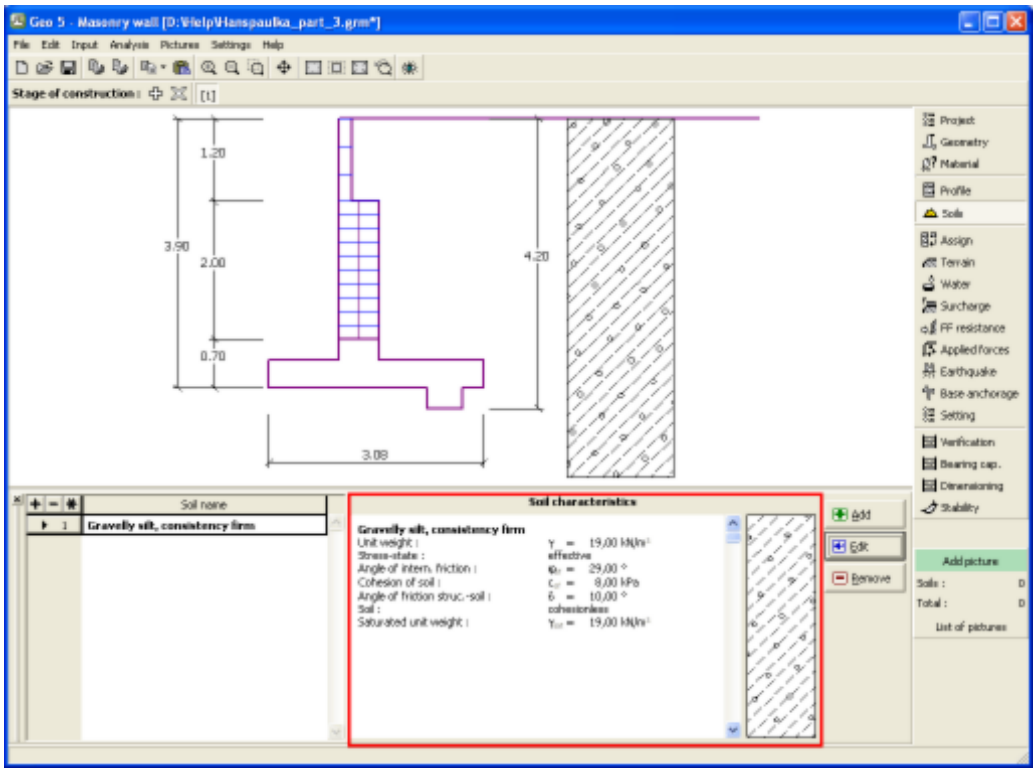
Frame "Profile"

Soils

The "Soils" frame contains a table with a list of inputted soils. The table also provides information about currently selected soil displayed in the right part of the frame.

Adding (editing) a soil is performed in the "Add (edit) soil" dialogue window.

The soil characteristics are specified in the program "Cantilever wall". These characteristics are further specified in chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

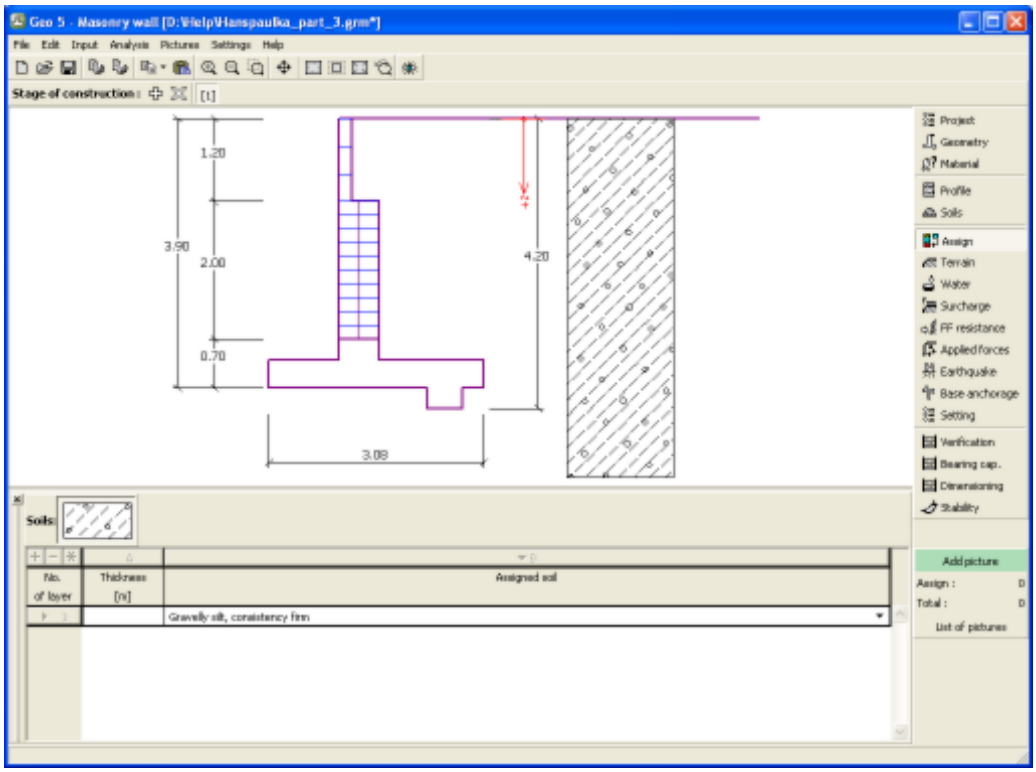


Frame "Soils"

Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

Procedure to assign soil into a layer is described in details herein.



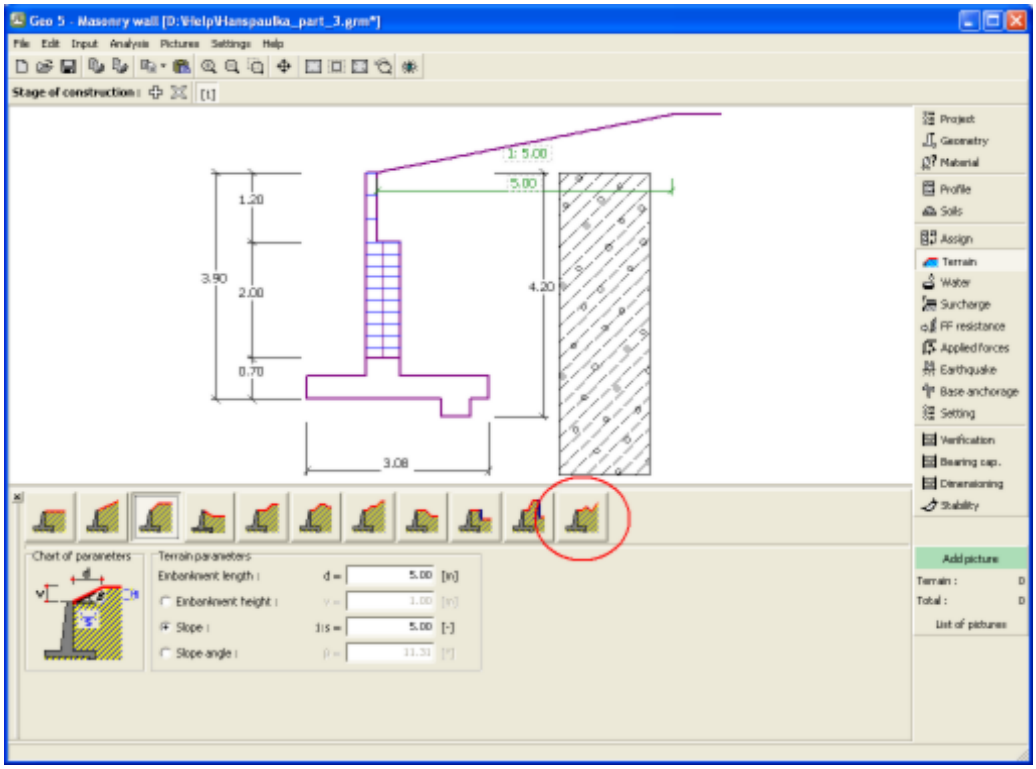
Frame "Assign"

Terrain

The "Terrain" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0, 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the hint, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

The "Water" allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. Water parameters (h1, h2...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

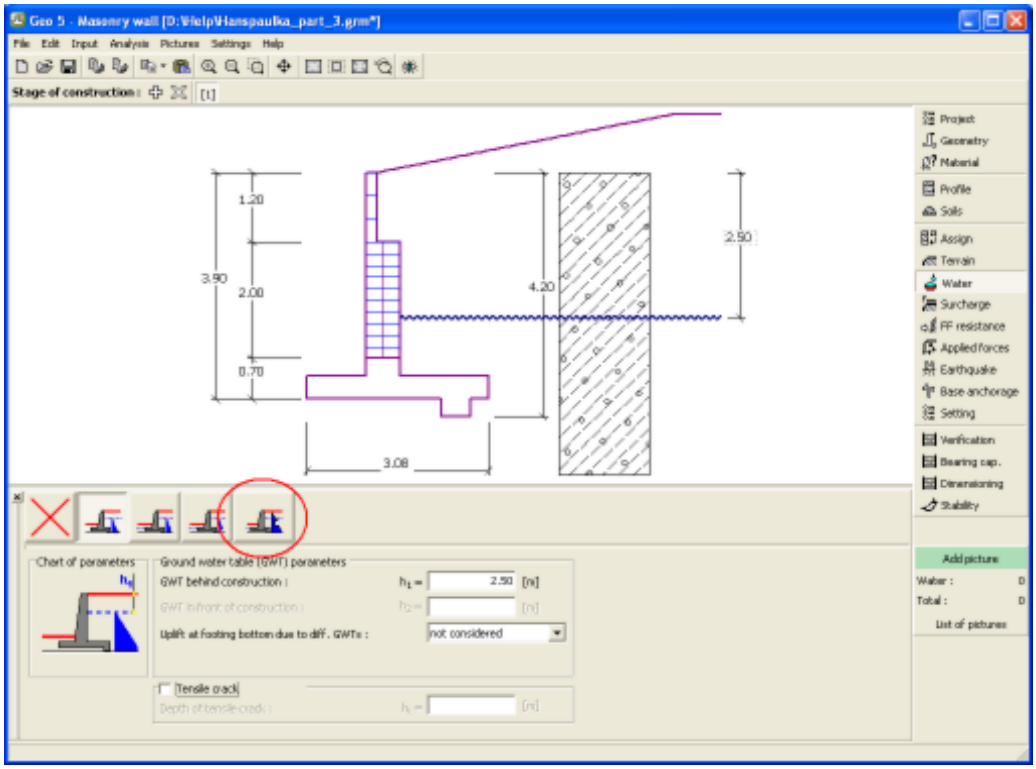
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic or it may not be considered at all. When verifying the wall, the uplift pressure in base of footing joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tab sheets "In front of structure" and "Behind structure" appear with tables. The table is filled with values of pore pressure in front, or behind the structure at a depth of "z" (z-axis).

The ground water table can also be specified above the structure or earth profile, respectively – in such a case the depth of water is inputted with negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the hint chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



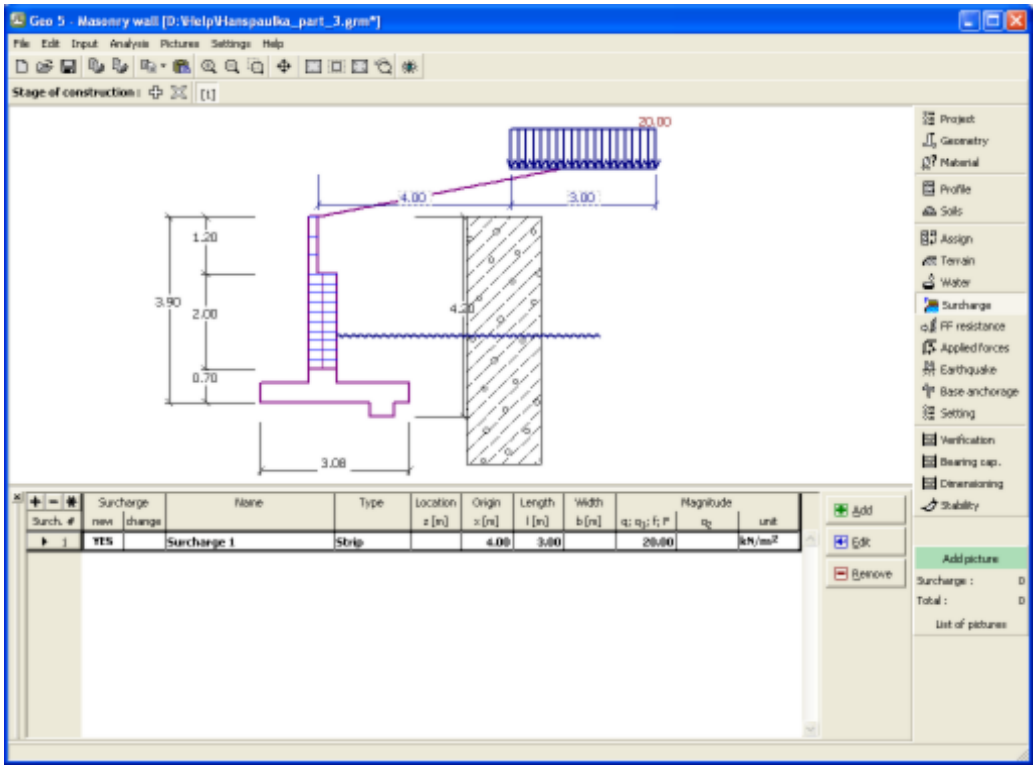
Frame "Water"

Surcharge

The "Surcharge" frame contains a table with a list of inputted surcharges. Adding (editing) surcharge is performed in the "New (edit) surcharge" dialogue window. The inputted surchages can be edited on the desktop with the help of active dimensions or active objects, respectively.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. Providing the surcharge is found off the terrain the computer prompts an error message.

Analysis of earth pressures due to surcharges is described in the theoretical part of the hint, chapter "Influence of surcharge".



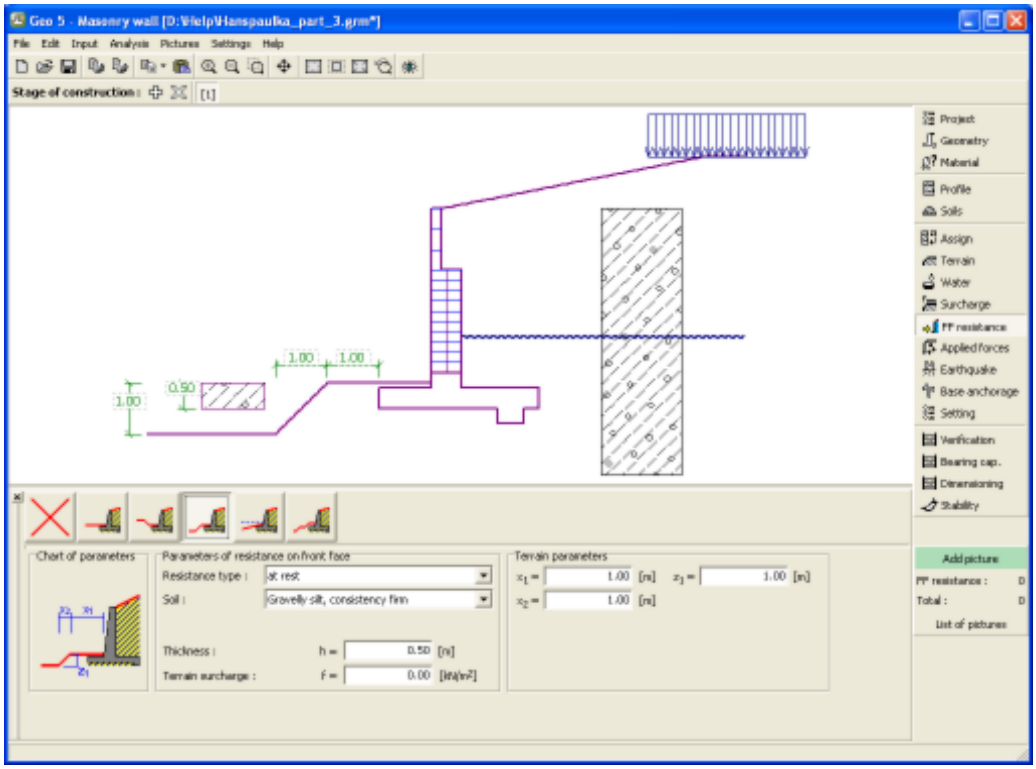
Frame "Surcharge"

Front face resistance

The "Front face resistance" frame allows by pressing the button for specifying the terrain shape and parameters of front face resistance. The selected shape with a graphic hint (" Parameter chart") of inputted values are displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Combo lists in the frame allows the user to select the type of resistance and a soil (the c ombo list contains soils introduced in the regime "Soils"). The magnitude of terrain surcharge in front of the wall or soil thickness above the wall lowest points can also be specified in the frame.

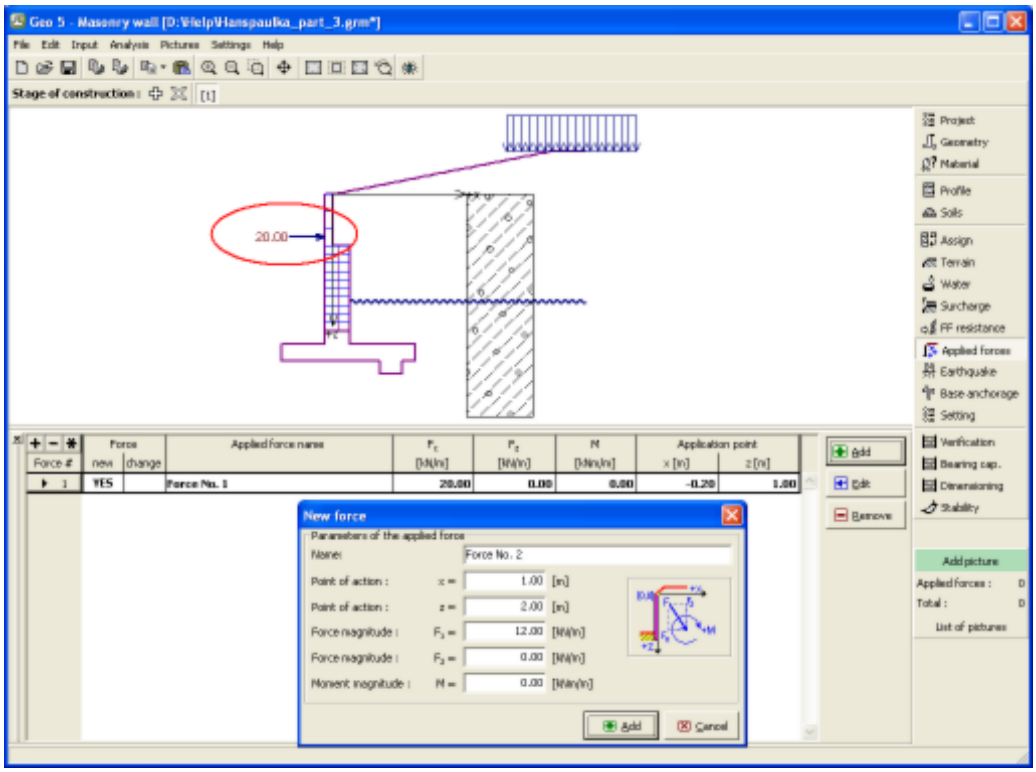
The resistance on a structure front face can be specified as a pressure at rest, passive pressure or reduced passive pressure. The resulting force due to reduced passive pressure is found as a resultant force caused by passive pressure multiplied by a corresponding coefficient, which follows from the inputted type of reduced passive pressure.



Frame "Front face resistance"

Inputted forces

The "Inputted forces" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "New force (edit force)" dialogue window. The inputted forces can also be edited on the desktop with the help of active objects.

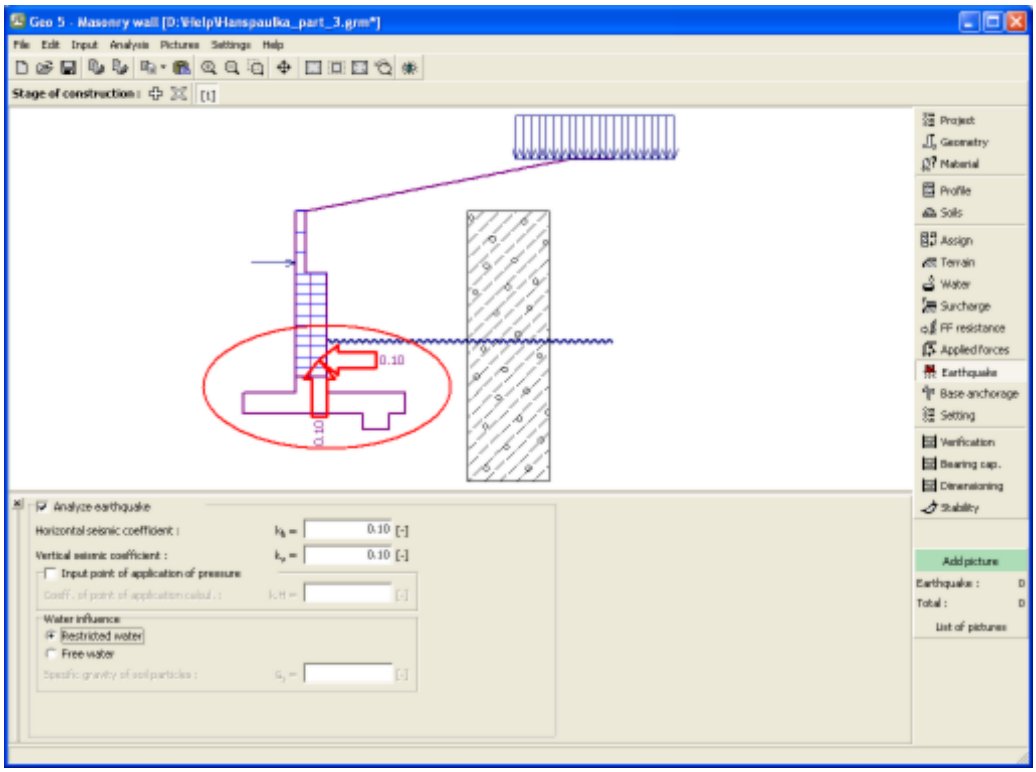


Frame "Inputted forces"

Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of inputted earthquake effects are displayed on the desktop.

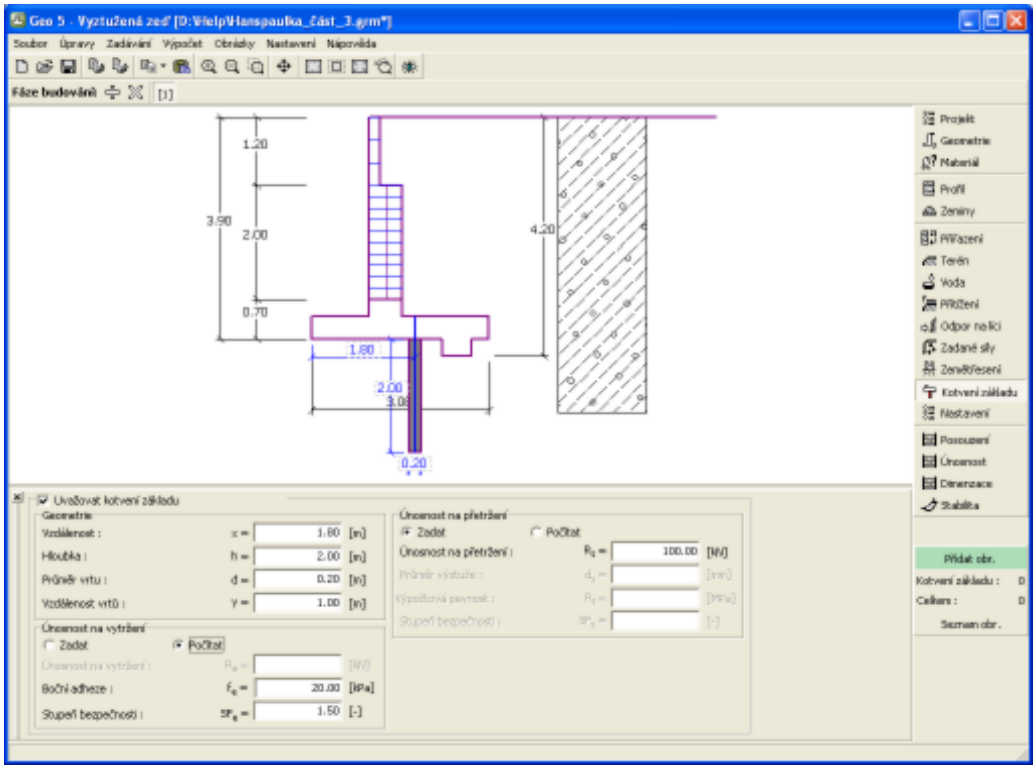
Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the hint in chapter "Influence earthquake".



Frame "Earthquake"

Base anchorage

The frame "Base anchorage" serves to input parameters (anchorage geometry, bearing capacity against pulling-out and pulling-apart) specifying an anchorage of the wall foundation. Geometry of footing anchorage can be edited either in the frame by inserting values in the inputting boxes or on the desktop with the help of active dimensions. The bearing capacity values can be either inputted or computed by the program from the inputted parameters.



Frame "Base anchorage"

Setting

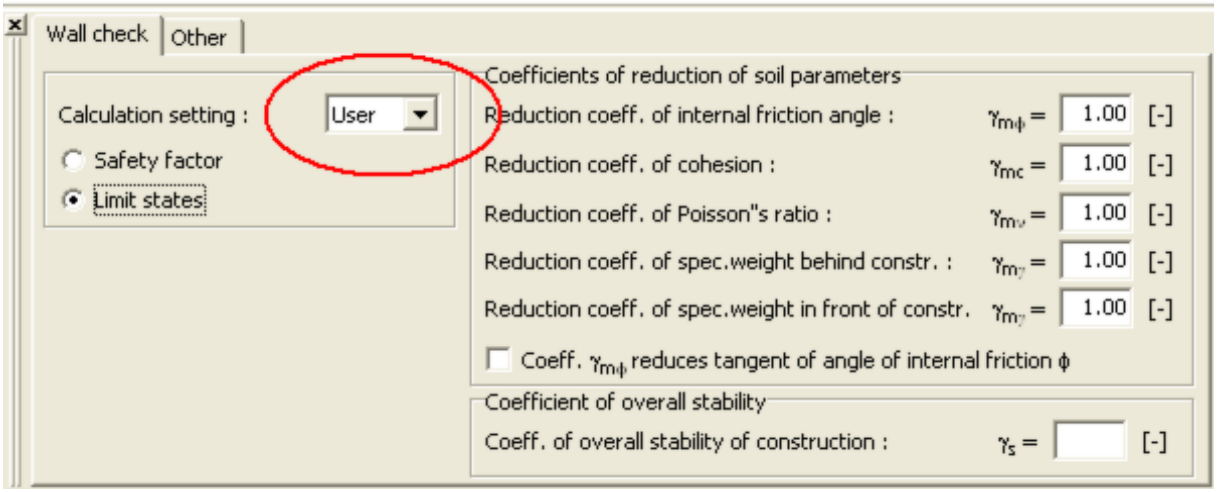
The "Setting" frame contains basic settings for the analysis of earth pressures. The program offers pre-setting for different countries (Czech Republic, Germany, France,...) and " Standard setting" recommended by the authors of the program for countries no included in the list. While changing settings in the combo list the values of coefficients of reduction of soil parameters in corresponding windows are changed.

Evaluating the structure according to theory of limit states also calls for the input of coefficient of overall stability of a structure. When subjecting the wall to overall verification this coefficient is used to multiply the resisting moment M_{res} and the resisting shear force H_{res}.

An arbitrary analysis setting is available with the option "User setting". Selecting the option "Reduce soil parameters" allows for specifying in input fields individual values of the coefficients of reduction of soil parameters (e.g., recommended values according to EC7-1).

The "Other" tab sheet serves to specify the type of pressure acting on a wall based on the allowable wall deformation. Providing the wall is free to move an active pressure is assumed, otherwise, a pressure at rest is used.

For wall with footing jump it is possible to choose in the tab sheet "Other" the way of accounting for the wall footing jump.


Wall check
Other
Calculation setting :
User
Safety factor
Limit states
Coefficients of reduction of soil parameters
Reduction coeff. of internal friction angle : $\gamma_{m\phi}$ = 1.00 [-]
Reduction coeff. of cohesion : γ_{mc} = 1.00 [-]
Reduction coeff. of Poisson"s ratio : $\gamma_{m\nu}$ = 1.00 [-]
Reduction coeff. of spec.weight behind constr. : $\gamma_{m\gamma}$ = 1.00 [-]
Reduction coeff. of spec.weight in front of constr. $\gamma_{m\gamma}$ = 1.00 [-]
Coeff. $\gamma_{m\phi}$ reduces tangent of angle of internal friction ϕ
Coefficient of overall stability
Coeff. of overall stability of construction : γ_s = [-]

Frame "Setting"

Verification

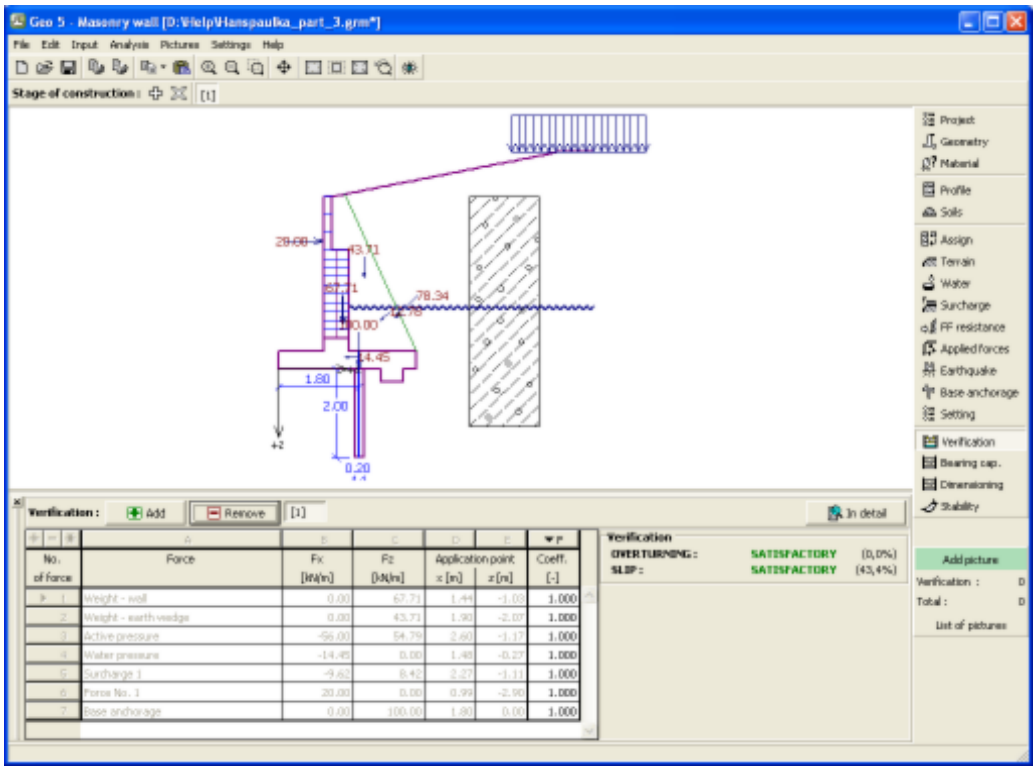
The "Verification" frame shows the analysis results. Several computation with different coefficients of resultant force effects can be carried out for a single task.

The wall is loaded either by active pressure or pressure at rest depending on input in the frame "Setting".

Procedure for wall verification is described in the theoretical part of the hint.

The computed forced are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against overturning and translation. The "In detail" button opens the dialogue window, which contains detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Verification"

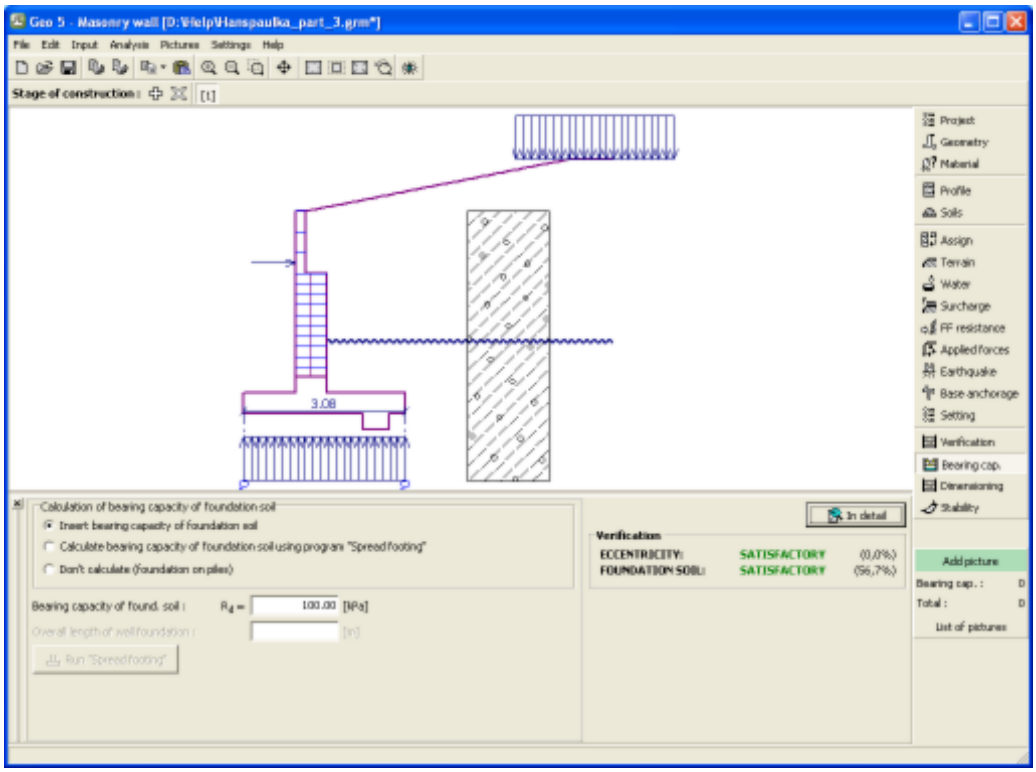
Bearing capacity

The "Bearing capacity" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "Verification". The program "Spread footing" then considers all verifications as loading cases.

Three basic analysis options are available in the frame:

- Input the foundation soil bearing capacity — The input field serves to specify the foundation soil bearing capacity. The results of verification analysis of a soil for eccentricity and bearing capacity are displayed in the right part of the frame. The "In detail" button opens the dialogue window that displays detailed listing of the results of verification analysis of foundation soil bearing capacity.
- Compute the foundation soil bearing capacity using the program "Spread footing" — Pressing the "Run "Spread footing" button starts the program "Spread footing" that allows for computing the soil bearing capacity or settlement and rotation of a footing. Pressing the "OK" button leaves the analysis regime – the results and all plots are copied to the program "Cantilever wall". The program "Spread footing" must be installed for the button to be active.
- Do not compute (pile footing) — The foundation soil bearing capacity is not computed.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Bearing capacity"

Dimensioning

The "Dimensioning" frame serves to design and verify the reinforcement of wall cross-section – the cross-section subjected to dimensioning is selected in the combo list.

- Construction joint verification — the number of a joint between masonry blocks is inputted

- Wall jump verification — type of assumed stress acting in construction joint for verification (linear, constant) can be selected in the frame "Project"

The wall is loaded either by the active earth pressure or by the pressure at rest depending on the setting in the frame "Setting".

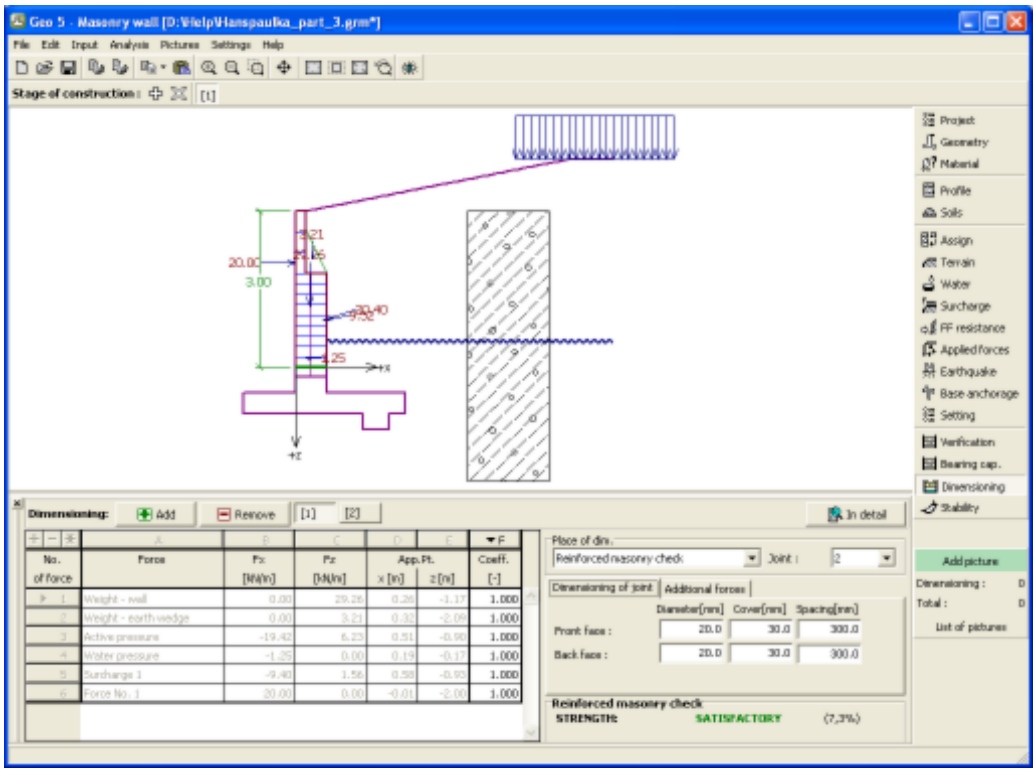
Procedure to derive distribution of internal forces in individual cross-sections is described in the theoretical part of this hint.

Joints between masonry blocks are verified according to AS 3700 "Masonry structures" standard. The program verifies the bearing capacity for bending, shear and combination of compression and bending. Reinforcement can be specified on both front and back sides of a structure. An additional loading applied to a cross-section (bending moment, compressive normal force and shear force) can also be specified. These additional forces are added to the computed ones.

Dimensioning of steel-reiforced concrete structure is performed according to the standard set in the frame "Project".

Several computations for various cross-sections can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "In detail" button opens the dialogue window that contains detailed listing of the dimensioning results.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.

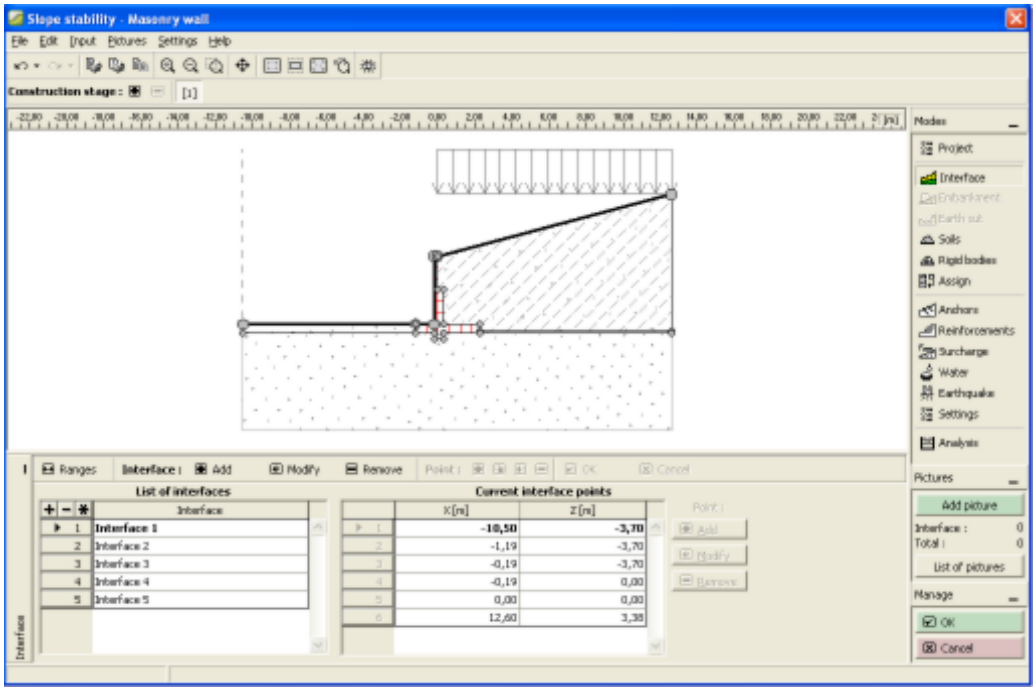


Frame "Dimensioning"

Stability

Pressing the "Stability" button launches the "Slope stability" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "Slope stability" is installed.

After completing all analyses press the "OK" button to leave the program – all data are then carried over to the analysis protocol of the "Masonry wall" program.



|*Frame "Stability"*

Program Gravity Wall

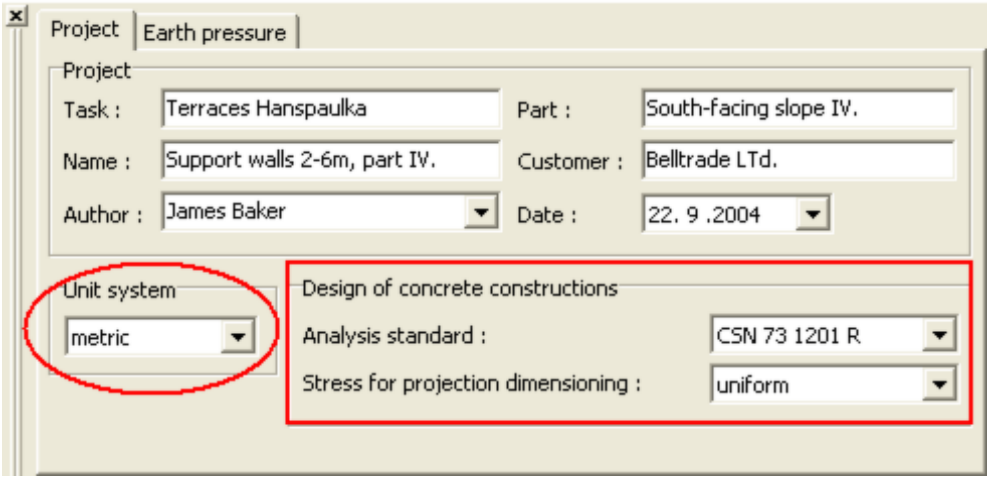
Project

The "Project" frame is used to input the basic project data and to specify the overall setting of the analysis run. The "Project" tab sheet contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc.

The "Project" tab sheet also allows the user to switch analysis units (metric / imperial).

The "Project" tab sheet further serves to specify a standard fornormy se concrete structures dimensioning. Referring to the selected standard the types of concrete and steel are then inputted in the "Material" frame. The dimensioning of cross-sections of the analyzed structure is then performed in the "Dimensioning" frame. Only ACI standard is available when selecting imperial units. While selecting standard it is also necessary to specify whether the stress in the foundation joint used for verification of a wall front key is assumed uniform (ČSN) or trapezoidal (Eurocode).

The "Earth pressures" tab sheet serves to choose the basic theory or standards to be followed in the solution of a given problem.

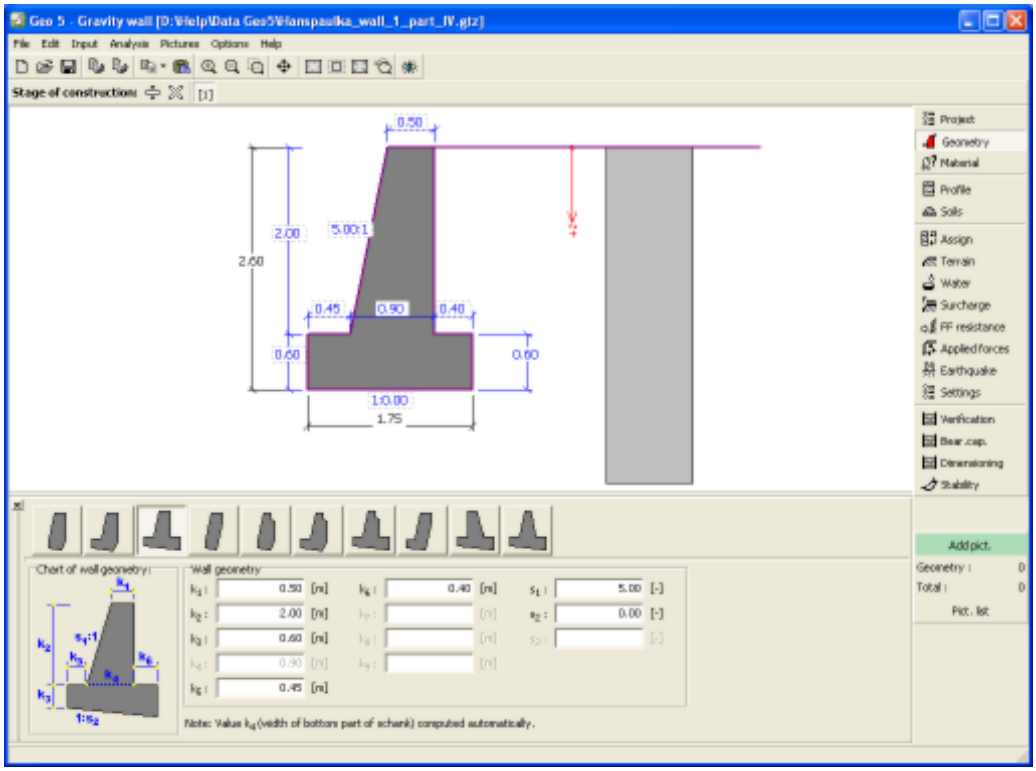


Frame Project - tab sheet "Project"

Geometry

The "Geometry" frame allows by pressing the button for selecting the wall shape. The selected shape with a graphic hint ("Wall geometry chart") appears in the left part of the frame. The shape of a wall can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

In case the structure is composed of inclined segments it is required to enter the ratio of sides of an inclined segment 1:x. The straight structure is specified by entering the value zero.



Frame "Geometry"

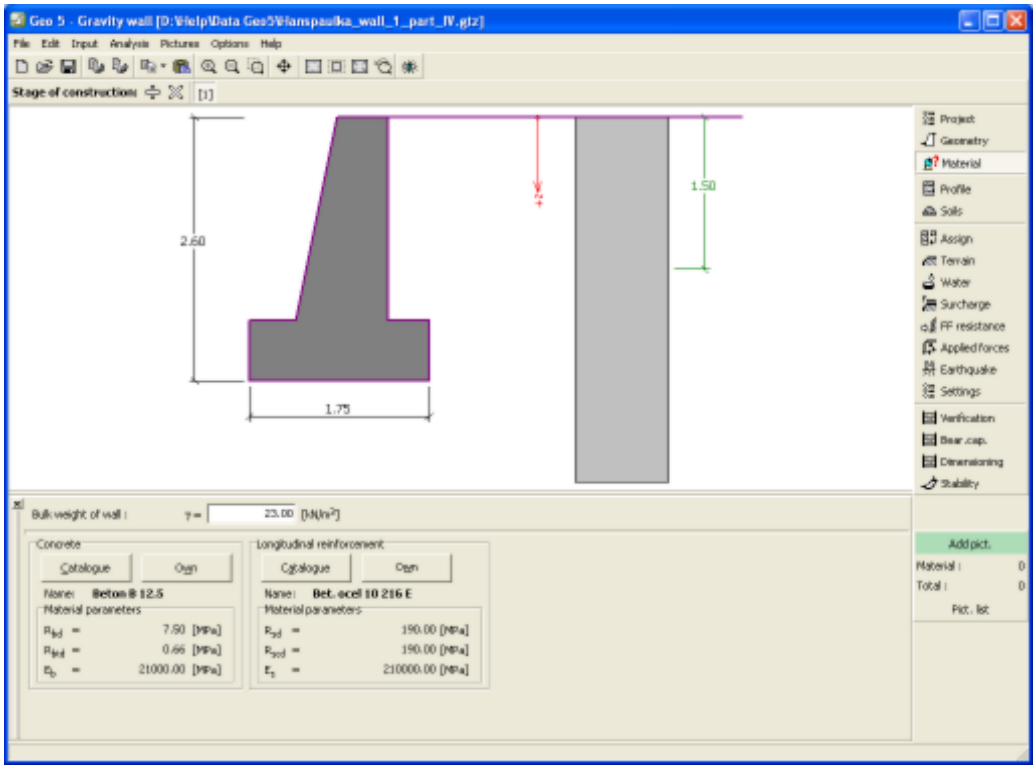
Material

The "Material" frame allows for the selection of material parameters for concrete and longitudinal steel reinforcements.

Two options are available when selecting the material type:

- the "Catalogue" button opens the "Material catalogue" dialogue window (for concrete or steel reinforcements), the list of materials then serves to select the desired material
- the "User" button opens the "Edit material – concrete" dialogue window (for concrete) or the "Edit material – concrete steel" dialogue window (for longitudinal steel reinforcements), which allows for manual specification of material parameters

The catalogues content depends on the selection of standard for the design of concrete structures set in the "Project" frame. The input field in the upper part of the frame serves to specify the wall bulk weight.



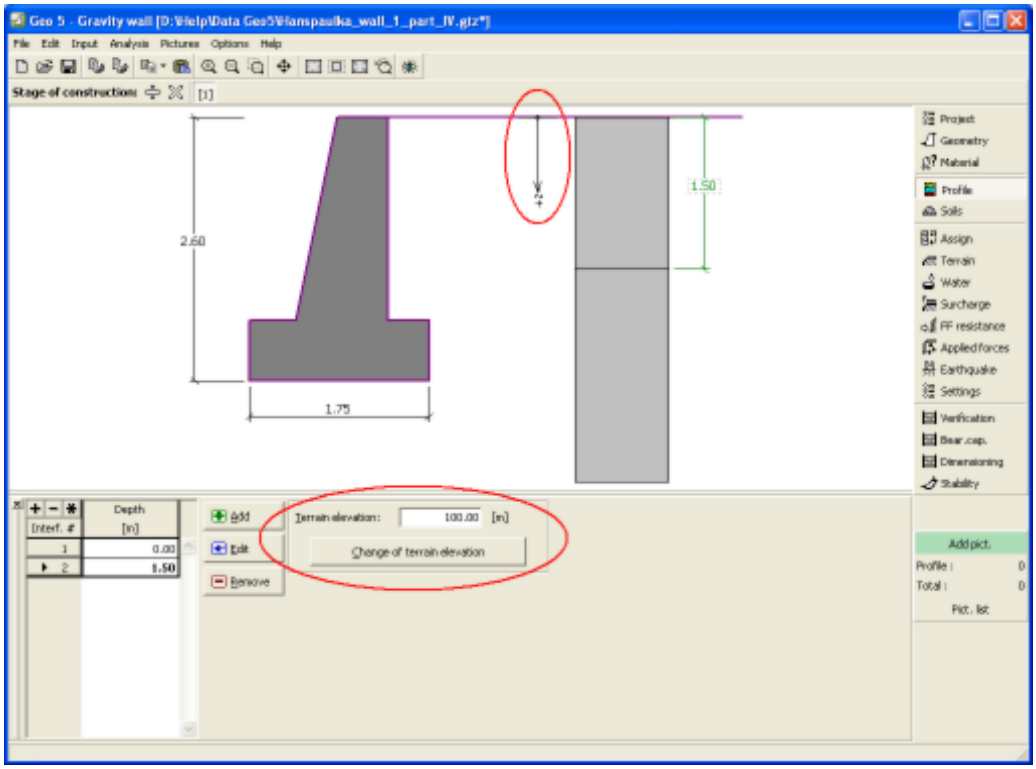
Frame "Material"

Profile

The "Profile" frame contains a table with a list of inputted interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "Add (edit) interface" dialogue window. The z-coordinate measured from the top point of a structure is specified.

The program allows for raising or lowering the top point of a structure in the "Change terrain elevation" dialogue window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".



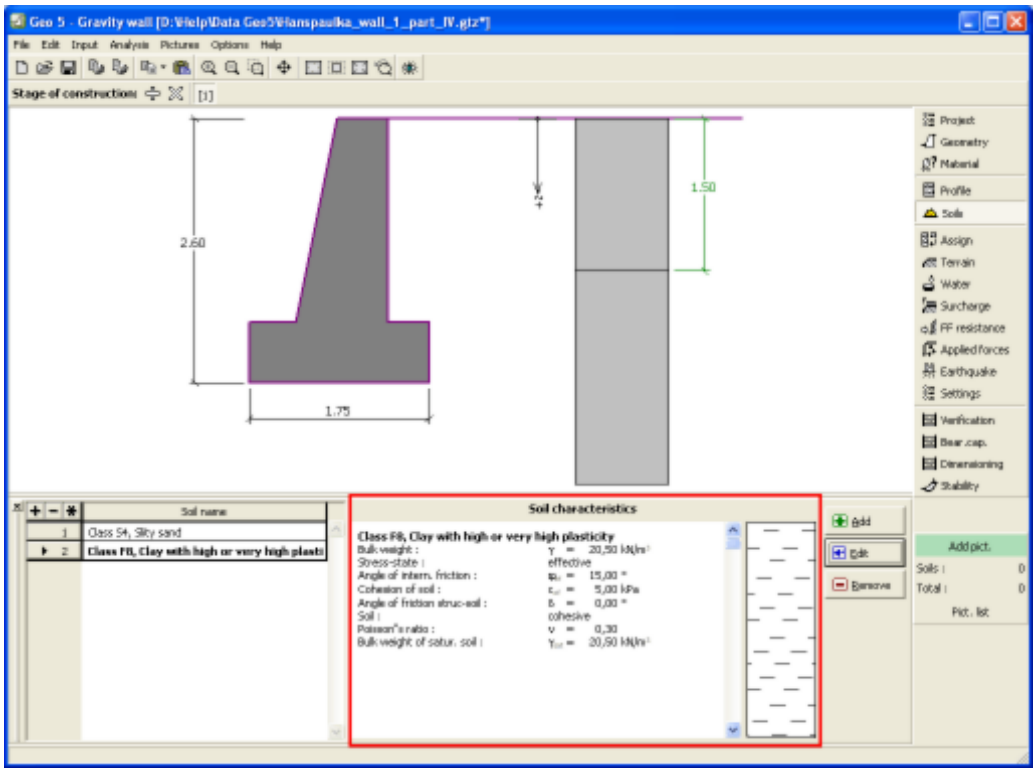
The frame "Profile"

Soils

The "Soils" frame contains a table with a list of inputted soils. The table also provides information about currently selected soil displayed in the right part of the frame.

Adding (editing) a soil is performed in the "Add (edit) soil" dialogue window.

The soil characteristics are specified in the program "Gravity wall". These characteristics are further specified in chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

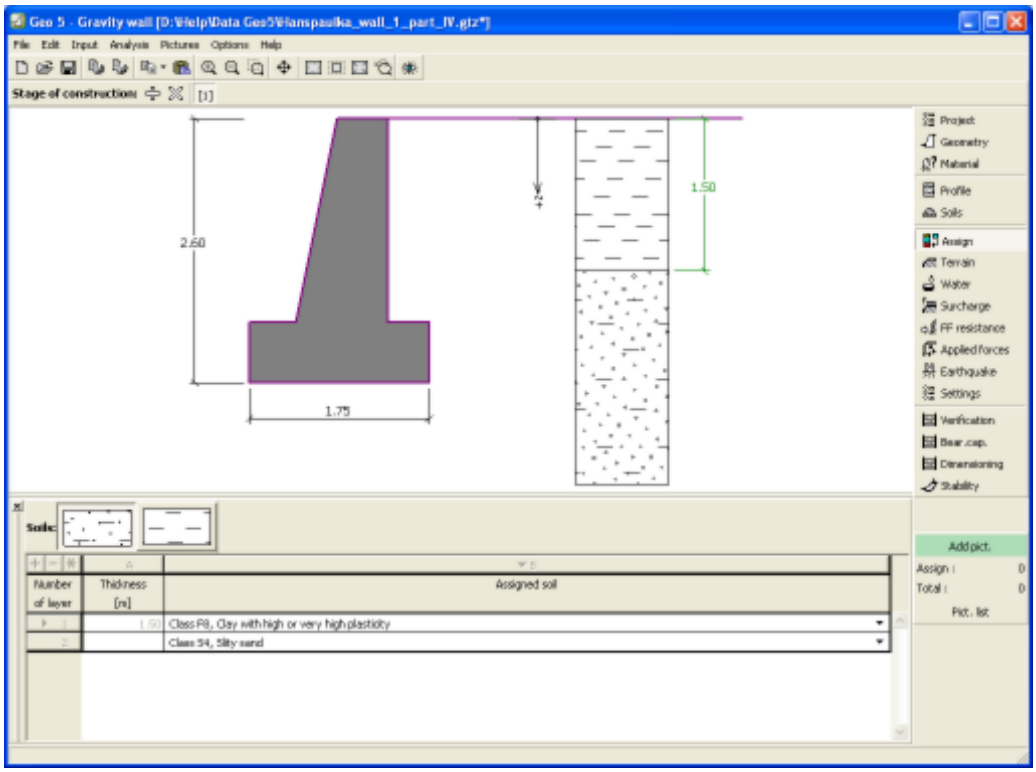


Frame "Soils"

Assign

The “Assign” frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

Procedure to assign soil into a layer is described in details herein.



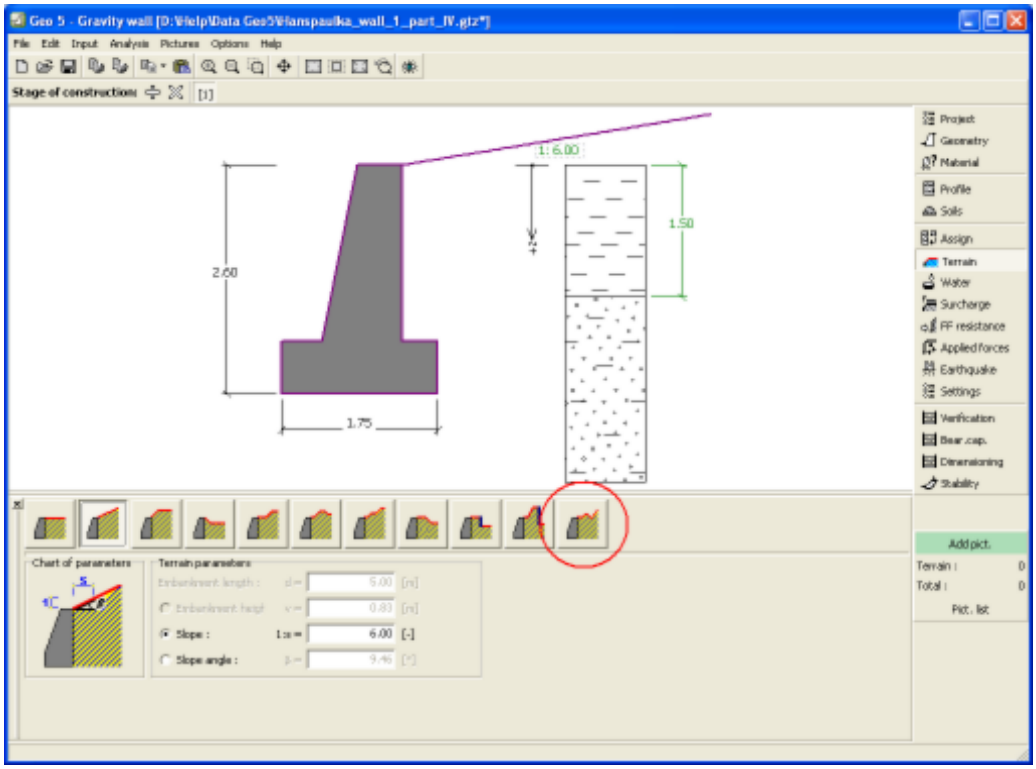
Frame "Assign"

Terrain

The "Terrain" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0, 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the hint, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

The "Water" allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. Water parameters (h1, h2...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

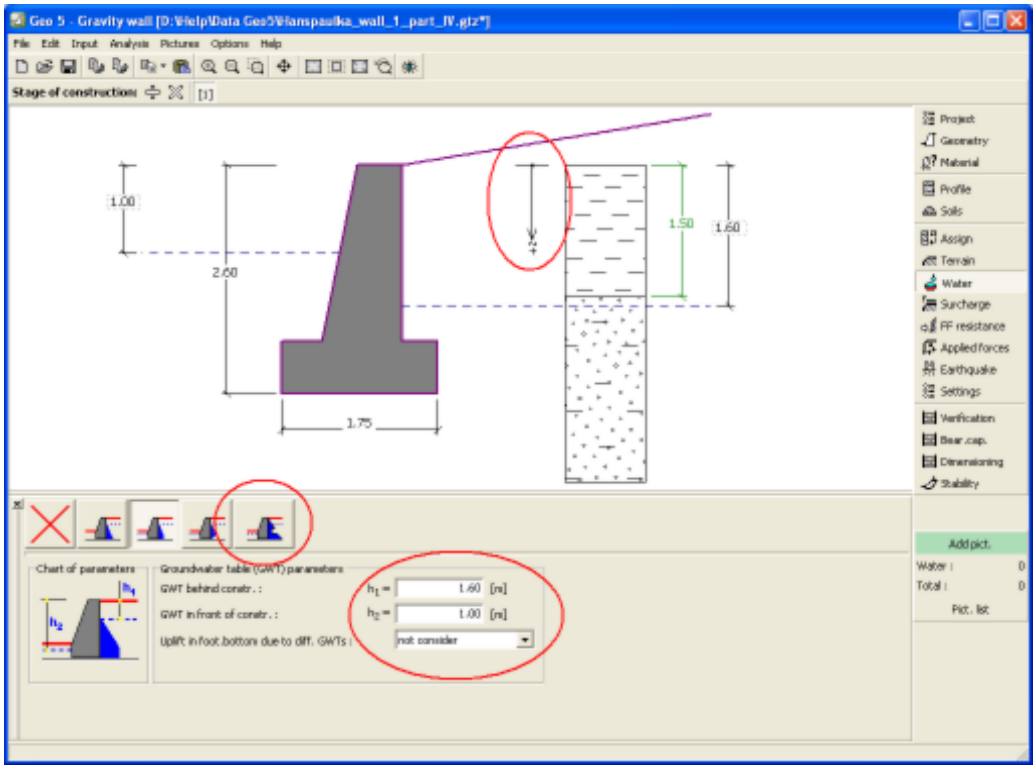
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic or it may not be considered at all. When verifying the wall, the uplift pressure in foundation joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tab sheets "In front of structure" and "Behind structure" appear with tables. The table is filled with values of pore pressure in front, or behind the structure at a depth of "z" (z-axis).

The ground water table can also be specified above the structure or earth profile, respectively – in such a case the depth of water is inputted with negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the hint chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



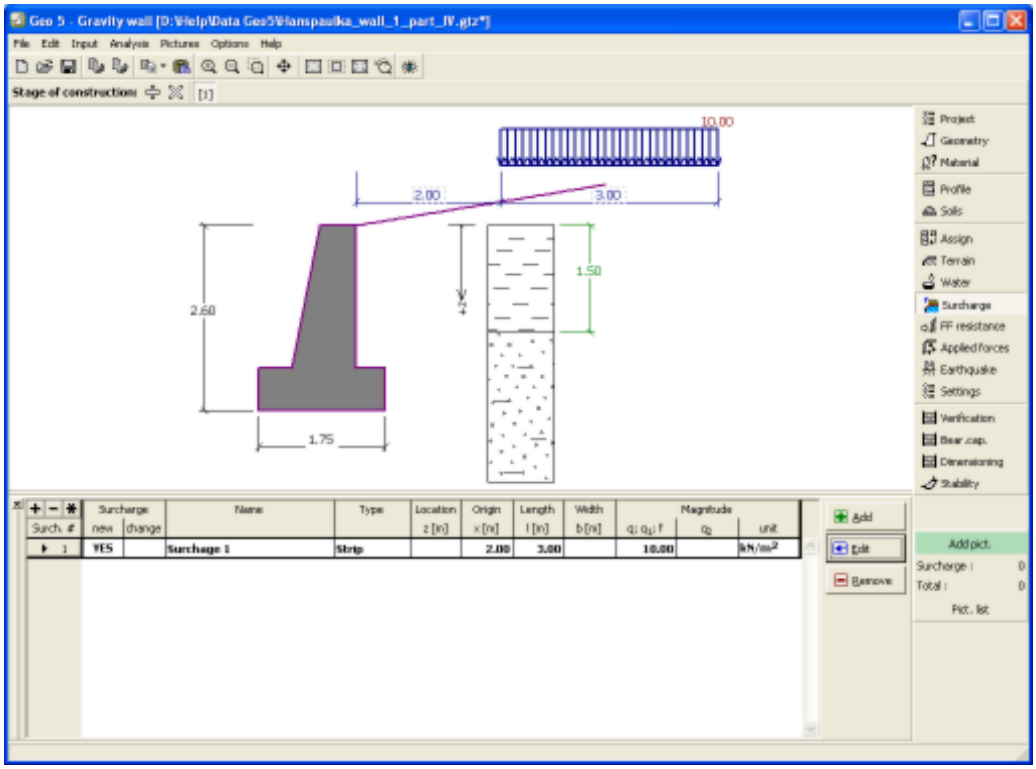
Frame "Water"

Surcharge

The "Surcharge" frame contains a table with a list of inputted surcharges. Adding (editing) surcharge is performed in the "New (edit) surcharge" dialogue window. The inputted surchages can be edited on the desktop with the help of active dimensions or active objects, respectively.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. Providing the surcharge is found off the terrain the computer prompts an error message.

Analysis of earth pressures due to surcharges is described in the theoretical part of the hint, chapter "Influence of surcharge".



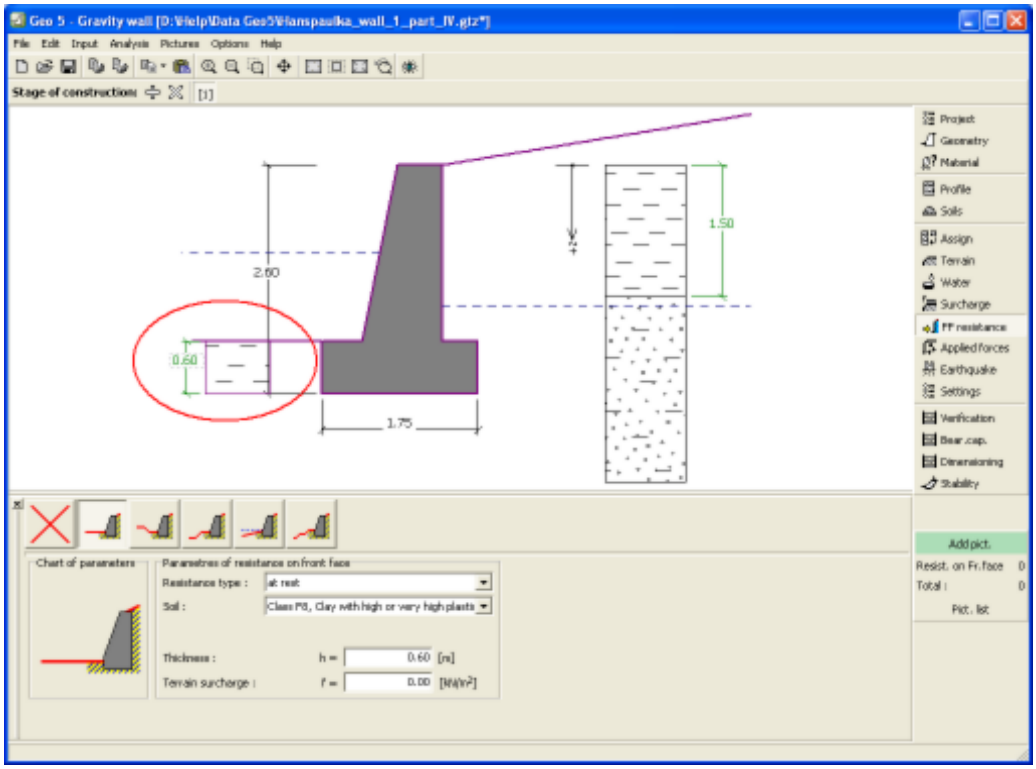
Frame "Surcharge"

Front face resistance

The "Front face resistance" frame allows by pressing the button for specifying theterrain shape and parameters of front face resistance. The selected shape with a graphic hint (" Parameter chart") of inputted values are displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Combo lists in the frame allows the user to select the type of resistance and a soil (the c ombo list contains soils introduced in the regime "Soils"). The magnitude of terrain surcharge in front of the wall or soil thickness above the wall lowest points can also be specified in the frame.

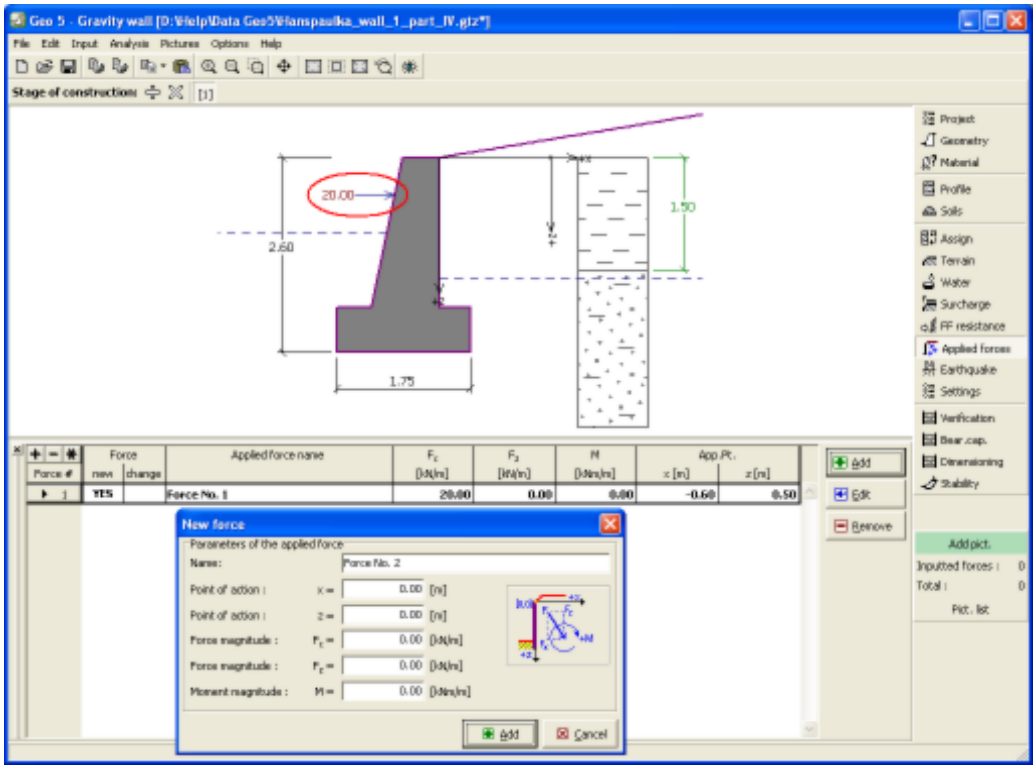
The resistance on a structure front face can be specified as a pressure at rest, passive pressure or reduced passive pressure. The resulting force due to reduced passive pressure is found as a resultant force caused by passive pressure multiplied by a corresponding coefficient, which follows from the inputted type of reduced passive pressure.



Frame "Front face resistance"

Inputted forces

The "Inputted forces" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "New force (edit force)" dialogue window. The inputted forces can also be edited on the desktop with the help of active objects.

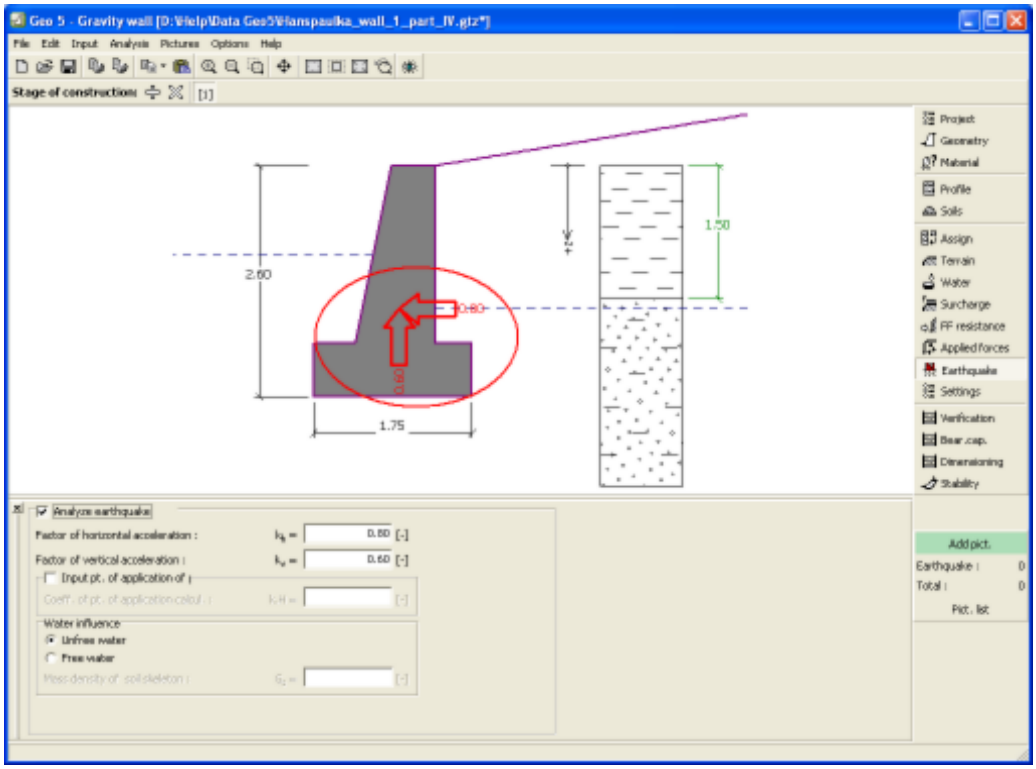


Frame "Inputted forces"

Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of inputted earthquake effects are displayed on the desktop.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the hint in chapter "Influence earthquake".



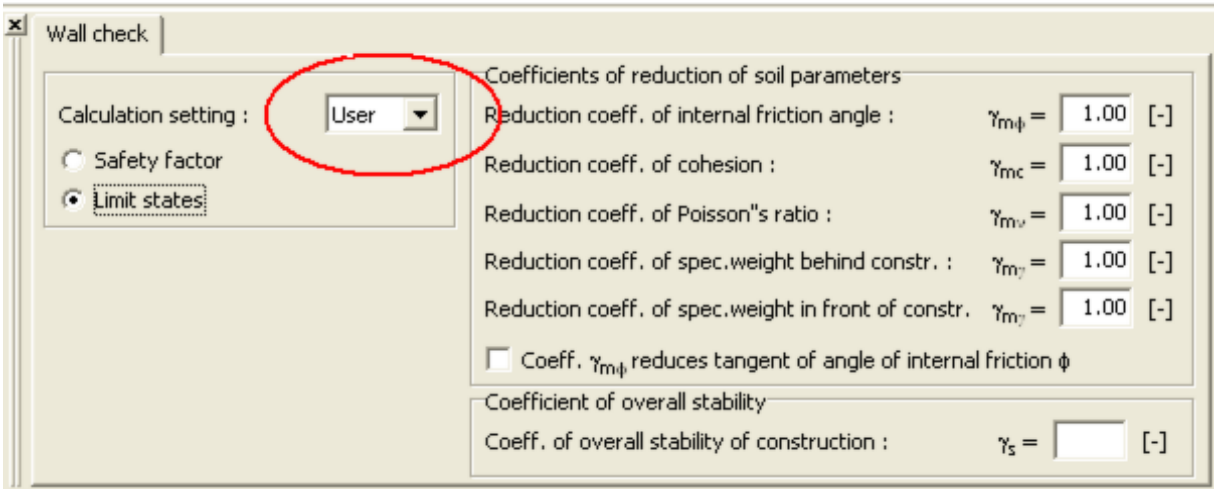
Frame "Earthquake"

Setting

The "Setting" frame contains basic settings for the analysis of earth pressures. The program offers pre-setting for different countries (Czech Republic, Germany, France,...) and " Standard setting" recommended by the authors of the program for countries no included in the list. While changing settings in the combo list the values of coefficients of reduction of soil parameters in corresponding windows are changed.

Evaluating the structure according to theory of limit states also calls for the input of coefficient of overall stability of a structure. When subjecting the wall to overall verification this coefficient is used to multiply the resisting moment M_{res} and the resisting shear force H_{res}.

An arbitrary analysis setting is available with the option "User setting". Selecting the option "Reduce soil parameters" allows for specifying in input fields individual values of the coefficients of reduction of soil parameters (e.g., recommended values according to EC7).


Wall check
Calculation setting :
User
Safety factor
Limit states
Coefficients of reduction of soil parameters
Reduction coeff. of internal friction angle : γmφ = 1.00 [-]
Reduction coeff. of cohesion : γmc = 1.00 [-]
Reduction coeff. of Poisson"s ratio : γmν = 1.00 [-]
Reduction coeff. of spec.weight behind constr. : γmγ = 1.00 [-]
Reduction coeff. of spec.weight in front of constr. γmγ = 1.00 [-]
Coeff. γmφ reduces tangent of angle of internal friction φ
Coefficient of overall stability
Coeff. of overall stability of construction : γs = [-]

Frame "Setting"

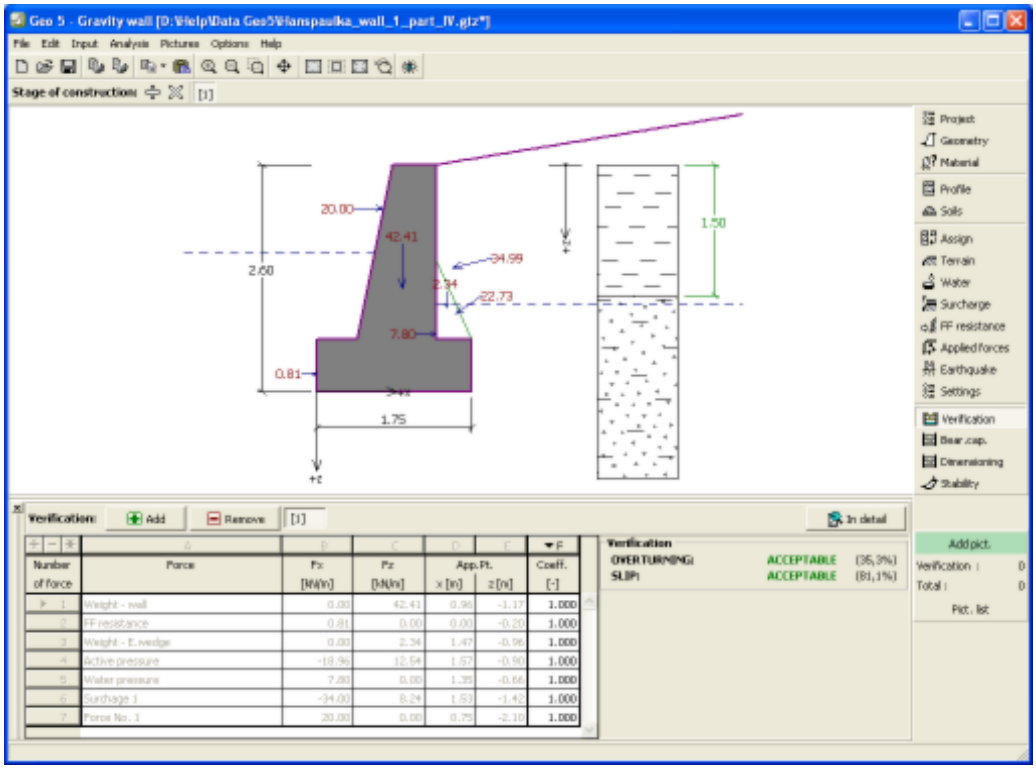
Verification

The "Verification" frame shows the analysis results. Several computation with different coefficients of resultant force effects can be carried out for a single task.

Procedure for wall verification is described in the theoretical part of the hint.

The computed forced are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against overturning and translation. The "In detail" button opens the dialogue window, which contains detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Verification"

Bearing capacity

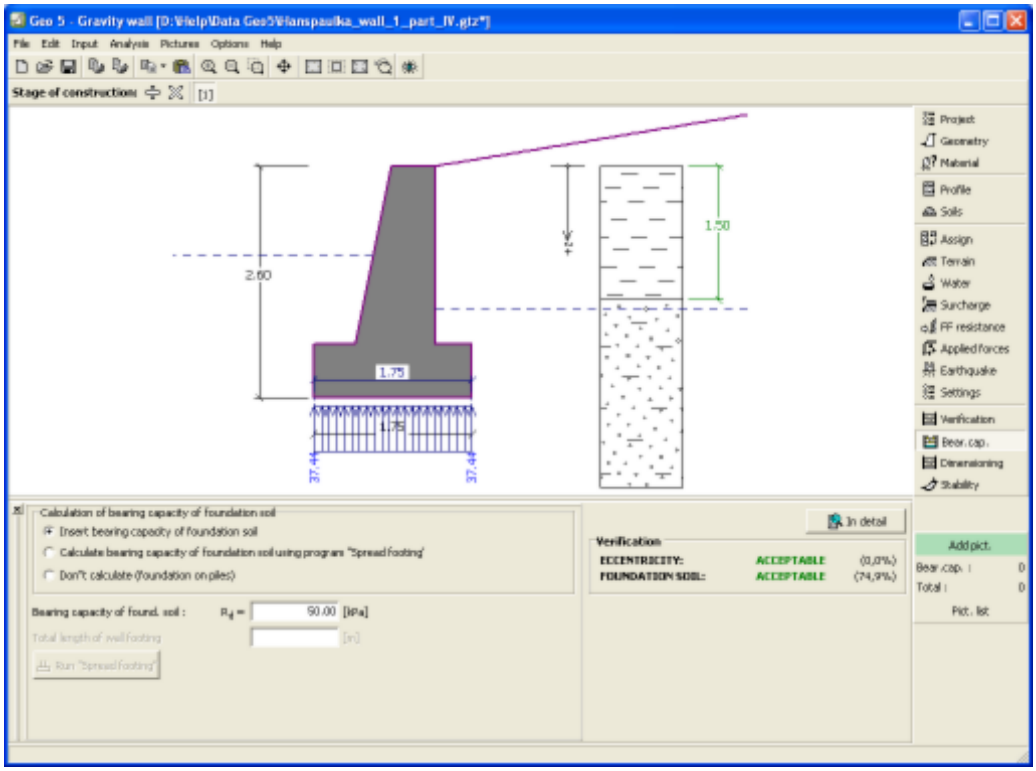
The "Bearing capacity" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all

verifications performed in the frame "Verification". The program "Spread footing" then considers all verifications as loading cases.

Three basic analysis options are available in the frame:

- Input the foundation soil bearing capacity — The input field serves to specify the foundation soil bearing capacity. The results of verification analysis of a soil for eccentricity and bearing capacity are displayed in the right part of the frame. The "In detail" button opens the dialogue window that displays detailed listing of the results of verification analysis of foundation soil bearing capacity.
- Compute the foundation soil bearing capacity using the program "Spread footing" — Pressing the "Run "Spread footing" button starts the program "Spread footing" that allows for computing the soil bearing capacity or settlement and rotation of a footing. Pressing the "OK" button leaves the analysis regime – the results and all plots are copied to the program "Gravity wall". The program "Spread footing" must be installed for the button to be active.
- Do not compute (pile footing) — The foundation soil bearing capacity is not computed.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Bearing capacity"

Dimensioning

The "Dimensioning" frame serves to design and verify the reinforcement of wall cross-section – the cross-section subjected to dimensioning is selected in the combo list.

- Wall stem verification
- Construction joint verification — depth of construction joint from construction top edge is specifiedse

- Wall jump verification — type of assumed stress acting in construction joint for verification (linear, constant) can be selected in the frame "Project"

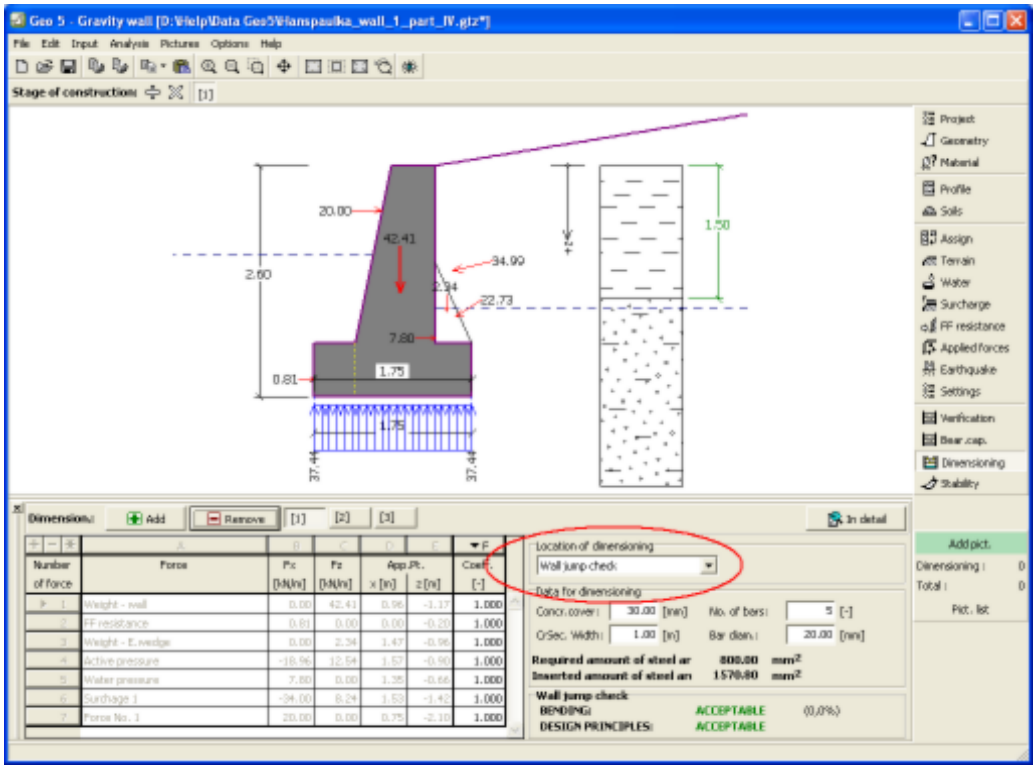
The wall stem and construction joint are always loaded by the pressure at rest. When verifying the front wall jump the wall is loaded either by the active pressure or the pressure at rest depending on input specified in the frame "Setting".

Procedure to derive distribution of internal forces in individual cross-sections is described in the theoretical part of this hint.

Dimensioning of the steel-reinforced concrete structure is performed according to the standard set in the frame "Project".

Several computations for various cross-sections can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "In detail" button opens the dialogue window that contains detailed listing of the dimensioning results.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.

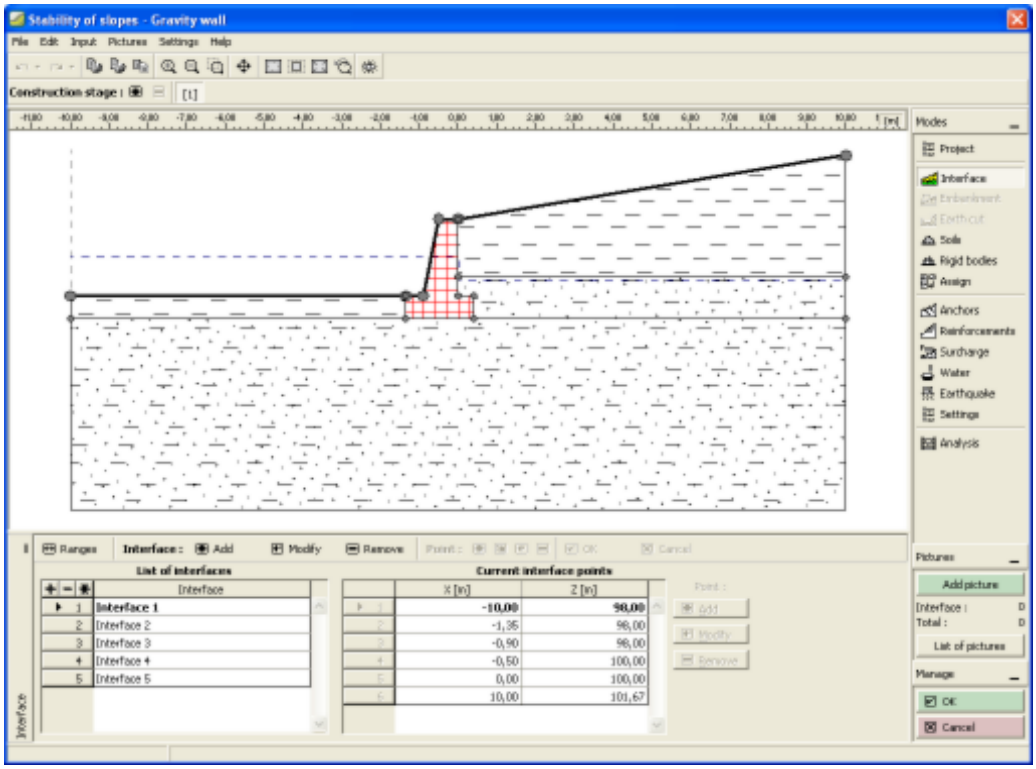


Frame "Dimensioning"

Stability

Pressing the "Stability" button launches the "Slope stability" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "Slope stability" is installed.

After completing all analyses press the "OK" button to leave the program – all data are then carried over to the analysis protocol of the "Gravity wall" program.



Frame "Stability"

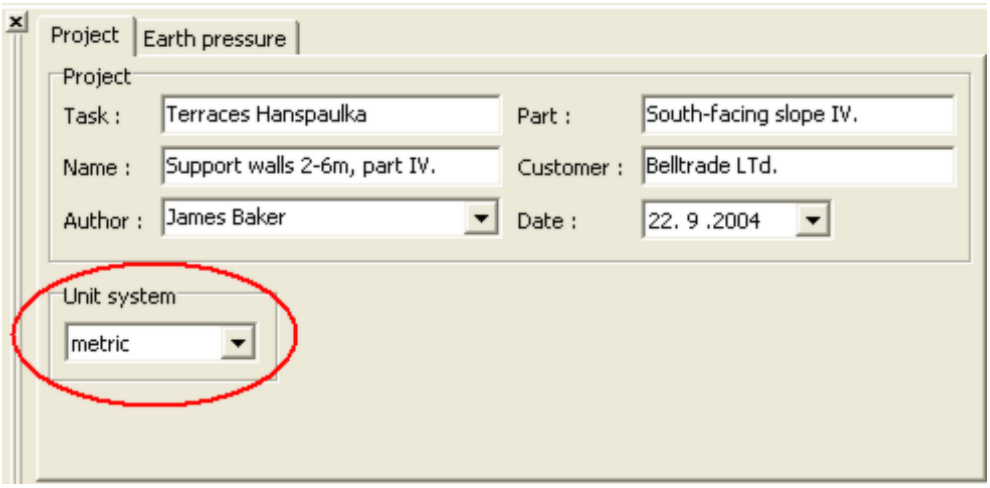
Program Block wall

Project

The "Project" frame is used to input the basic project data and to specify the overall setting of the analysis run. The "Project" tab sheet contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc.

The "Project" tab sheet also allows the user to switch analysis units (metric / imperial).

The "Earth pressures" tab sheet serves to choose the basic theory or standards to be followed in the solution of a given problem.



Frame "Project" - tab sheet "Project"

Geometry

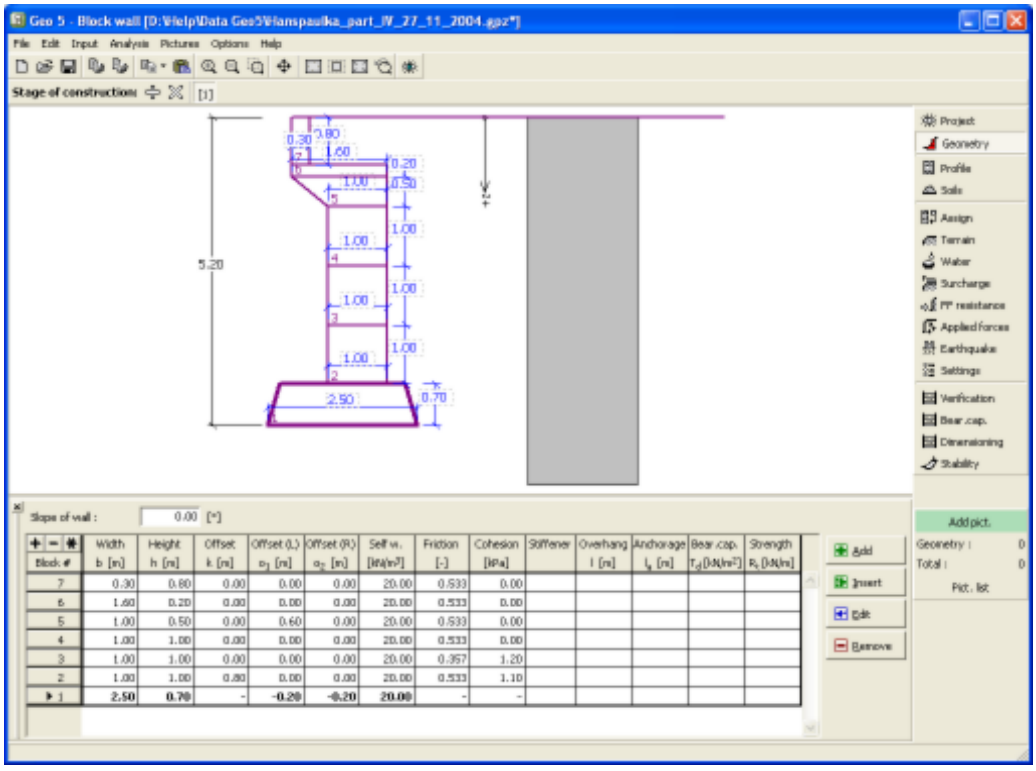
The "Geometry" frame contains a table with a list of inputted structural precast units

(blocks) of a wall (the lowest block is labeled as No. 1). Adding (editing) blocks is performed in the "New block (Edit block)" dialogue window.

This dialogue window serves to define the geometry of a block, parameters of reinforcement (overhang length, anchorage length, bearing capacity against pull out, reinforcement strength) and material characteristics (self weight, shear resistance between two blocks, cohesion).

The program allows for adding (inserting) another block in between two already existing blocks of a structure. Inserting a new block is performed in the "Insert block" dialog window that complies with the "New block" dialogue window. The inserted block is ordered such to proceed the currently selected block of a structure.

The inputted blocks can be further edited on the desktop with the help of active dimensions or active objects - double clicking on a structure opens a dialog window with a given block. When using the regime of active objects the visualization of detailed dimensions must be turned off in the "Setting visualization style" dialogue window.



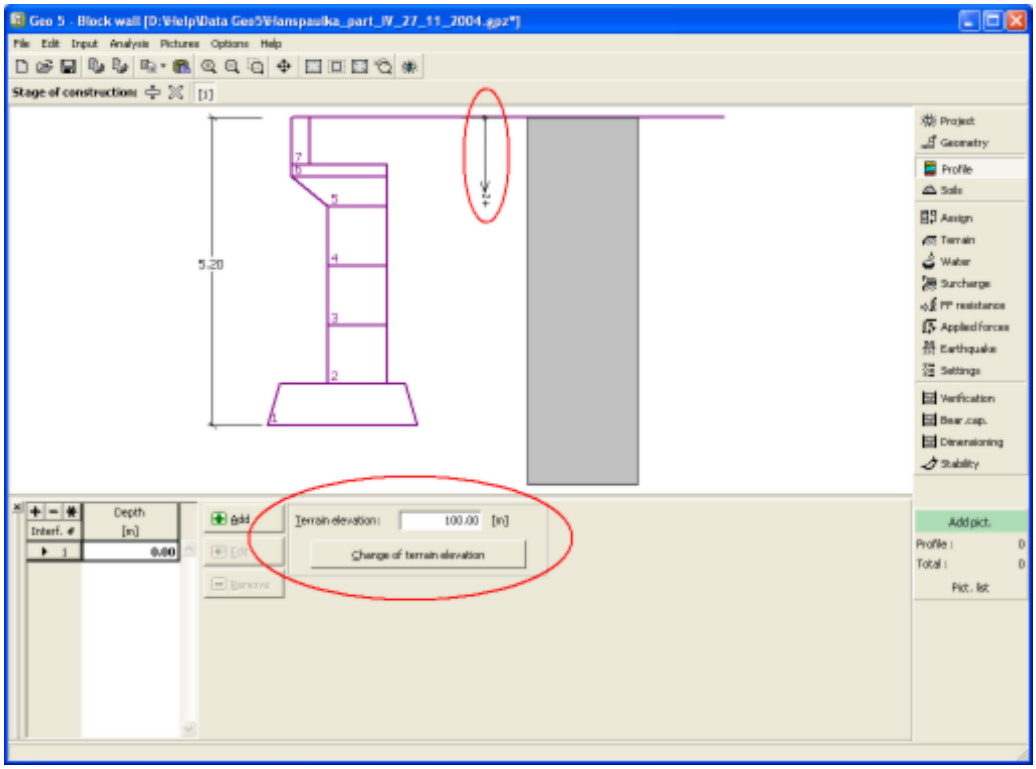
Frame "Geometry"

Profile

The "Profile" frame contains a table with a list of inputted interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "Add (edit) interface" dialogue window. The z-coordinate measured from the top point of a structure is specified.

The program allows for raising or lowering the top point of a structure in the "Change terrain elevation" dialogue window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".



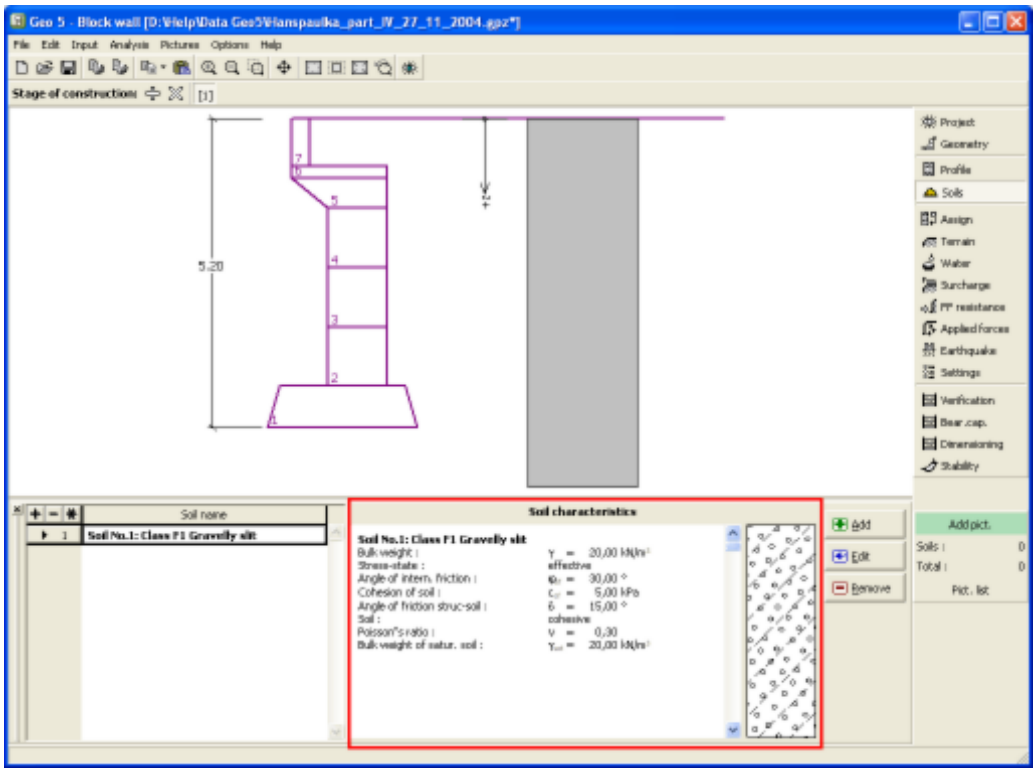
Frame "Profile"

Soils

The "Soils" frame contains a table with a list of inputted soils. The table also provides information about currently selected soil displayed in the right part of the frame.

Adding (editing) a soil is performed in the "Add (edit) soil" dialogue window.

The soil characteristics are specified in the program "Block wall". These characteristics are further specified in chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

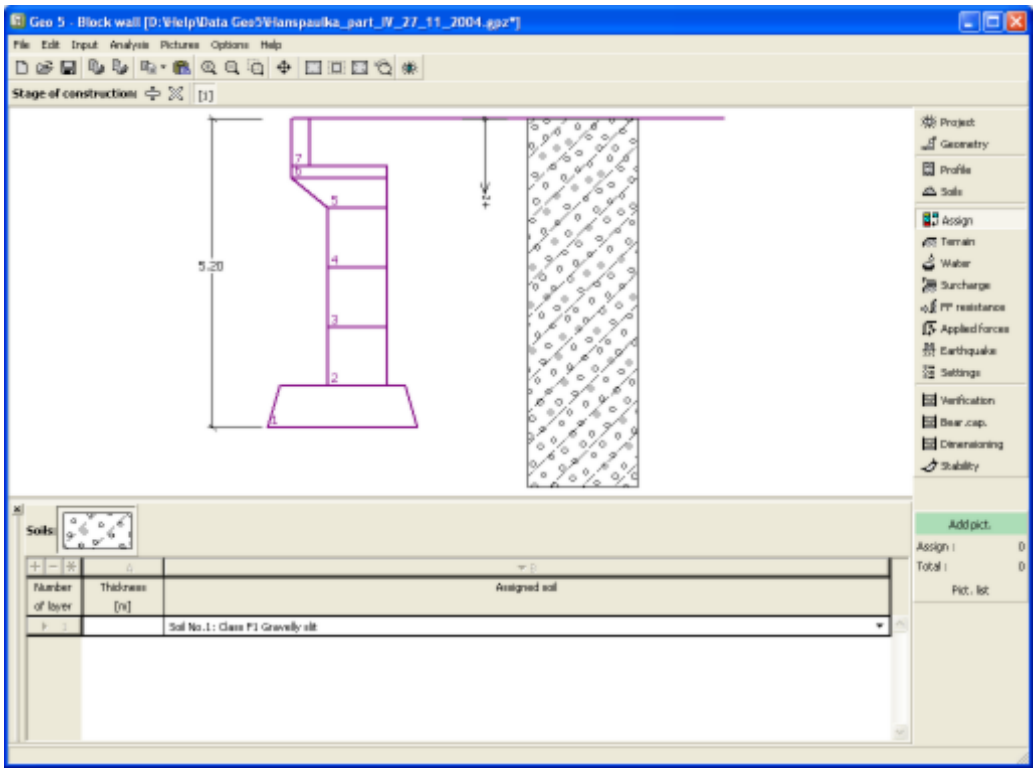


Frame "Soils"

Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

Procedure to assign soil into a layer is described in details herein.



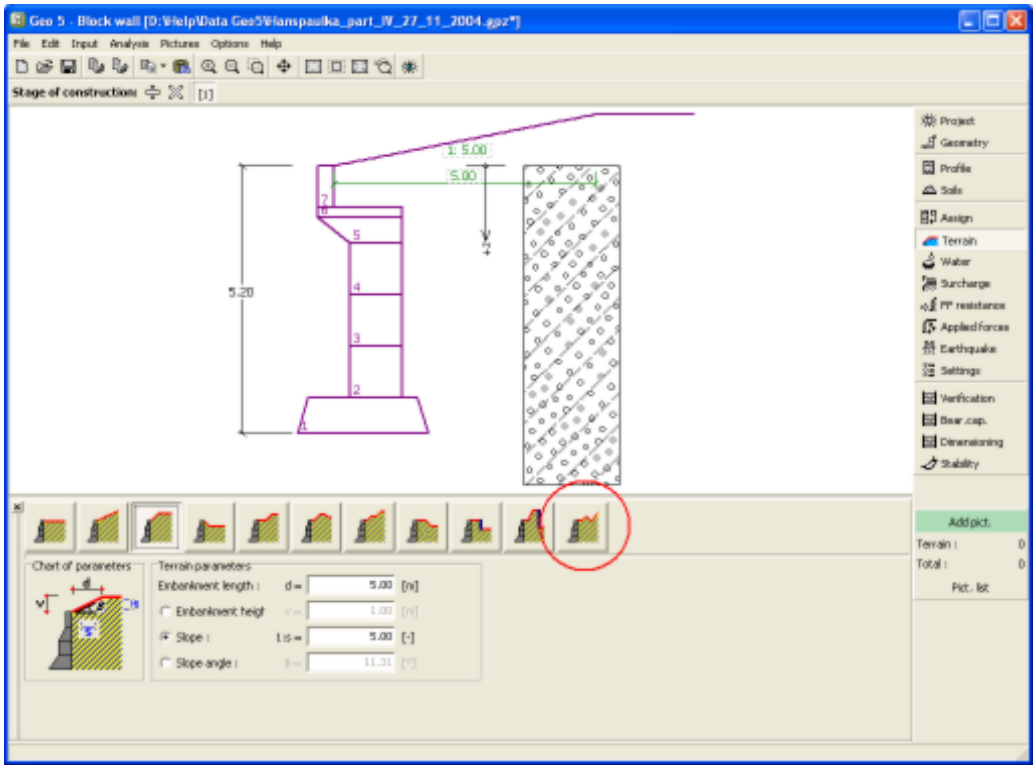
Frame "Assign"

Terrain

The "Terrain" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0, 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the hint, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

The "Water" frame allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. Water parameters h_1, h_2 can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

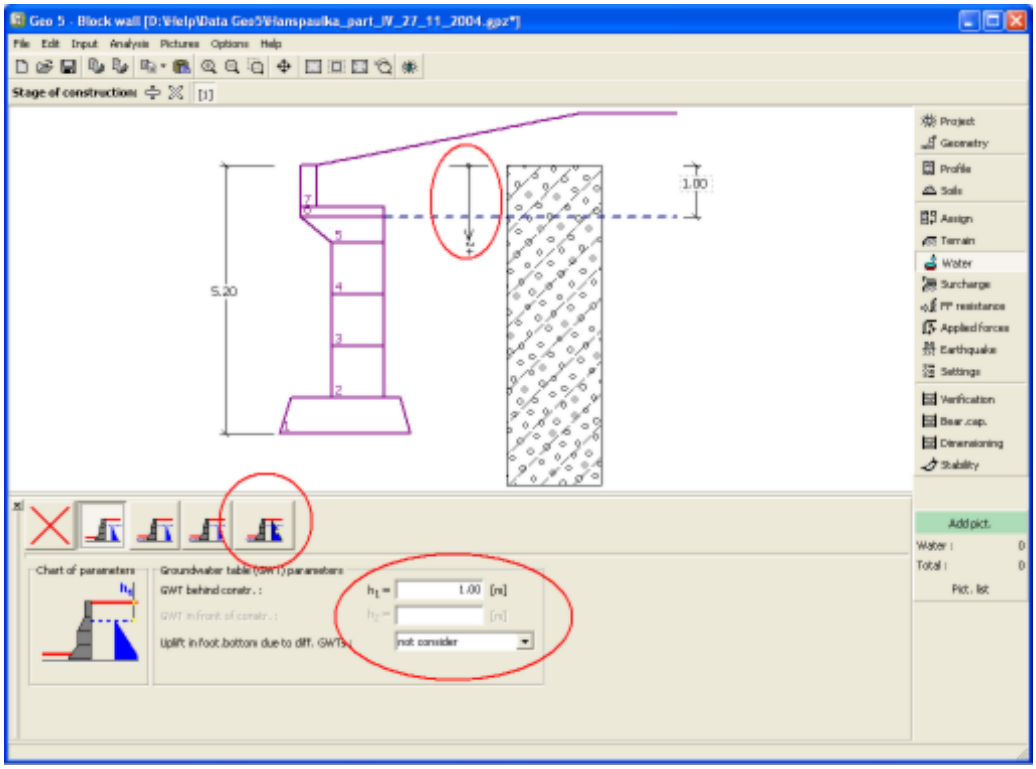
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic or it may not be considered at all. When verifying the wall, the uplift pressure in foundation joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tab sheets "In front of structure" and "Behind structure" appear with tables. The table is filled with values of pore pressure in front, or behind the structure at a depth of "z" (z-axis).

The ground water table can also be specified above the structure or earth profile, respectively – in such a case the depth of water is inputted with negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the hint chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



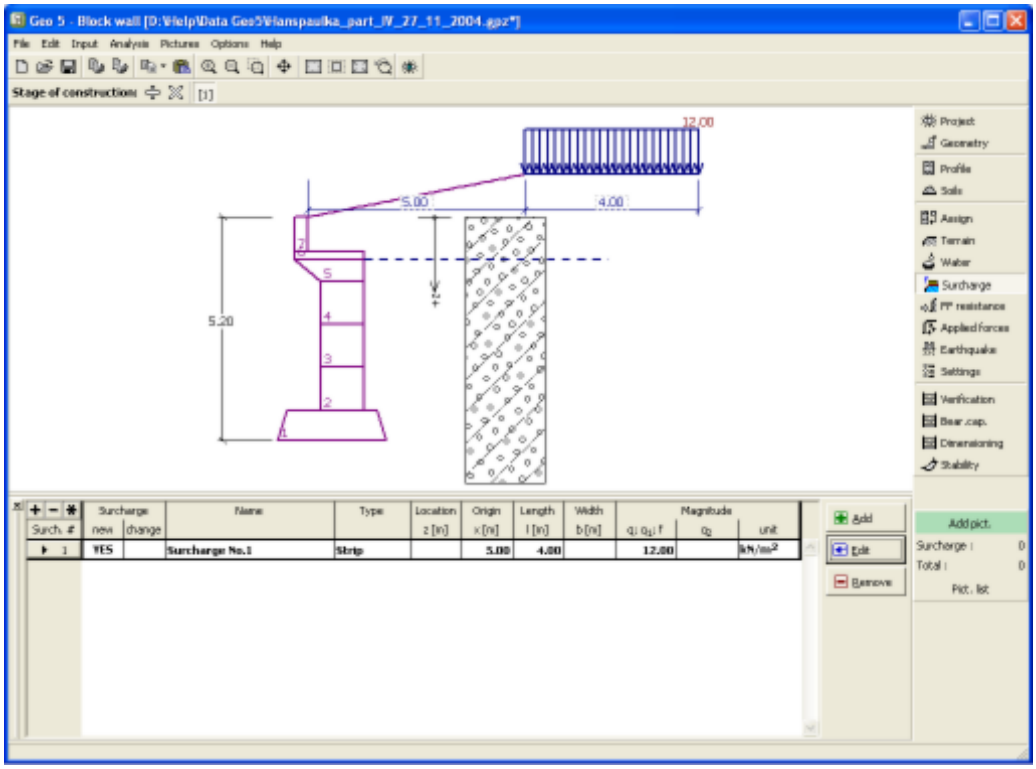
Frame "Water"

Surcharge

The "Surcharge" frame contains a table with a list of inputted surcharges. Adding (editing) surcharge is performed in the "New (edit) surcharge" dialogue window. The inputted surchages can be edited on the desktop with the help of active dimensions or active objects, respectively.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. Providing the surcharge is found off the terrain the computer prompts an error message.

Analysis of earth pressures due to surcharges is described in the theoretical part of the hint, chapter "Influence of surcharge".



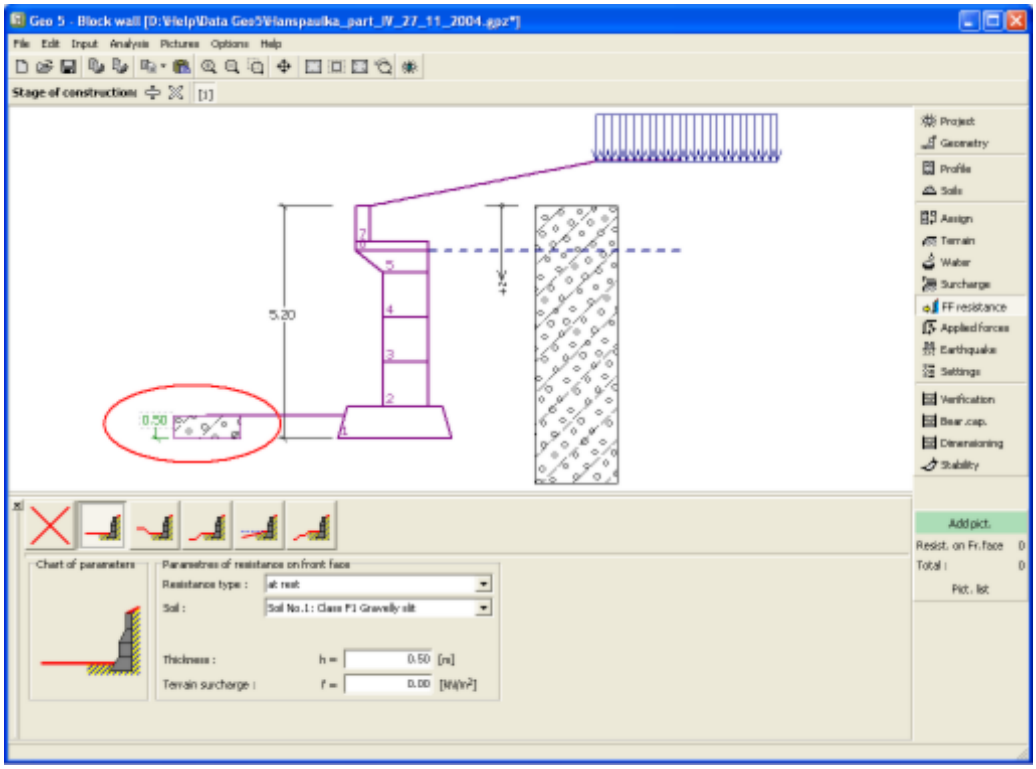
Frame "Surcharge"

Front face resistance

The "Front face resistance" frame allows by pressing the button for specifying the terrain shape and parameters of front face resistance. The selected shape with a graphic hint (" Parameter chart") of inputted values are displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Combo lists in the frame allows the user to select the type of resistance and a soil (the c ombo list contains soils introduced in the regime "Soils"). The magnitude of terrain surcharge in front of the wall or soil thickness above the wall lowest points can also be specified in the frame.

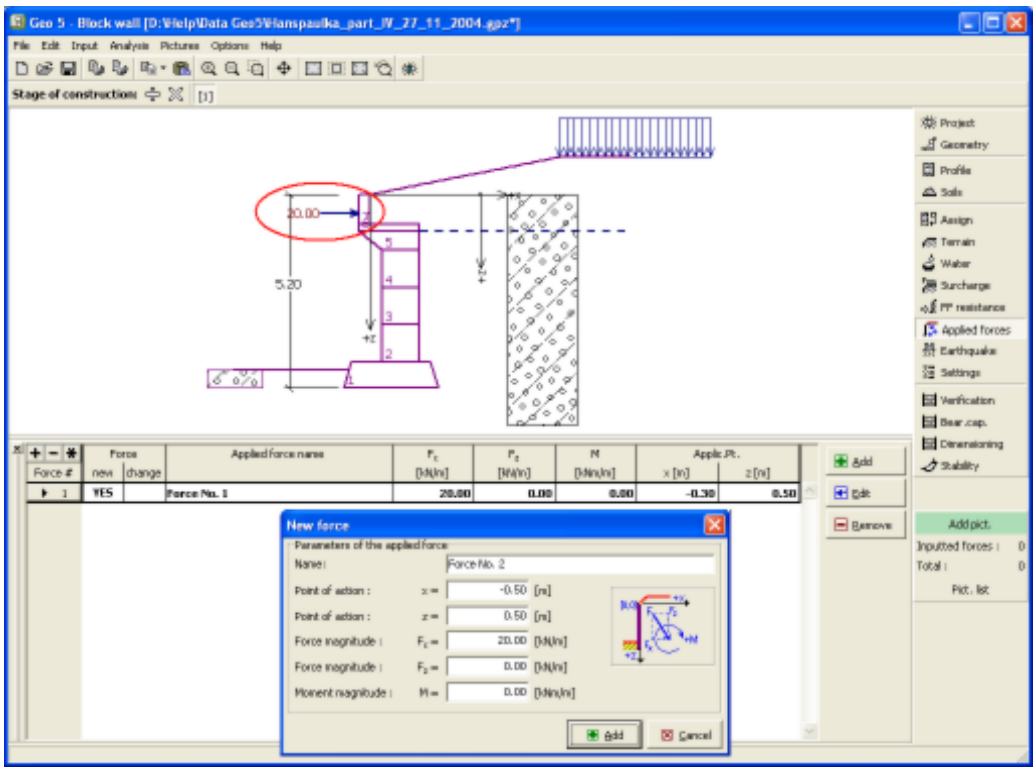
The resistance on a structure front face can be specified as a pressure at rest, passive pressure or reduced passive pressure. The resulting force due to reduced passive pressure is found as a resultant force caused by passive pressure multiplied by a corresponding coefficient, which follows from the inputted type of reduced passive pressure.



Frame "Front face resistance"

Inputted forces

The "Inputted forces" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "New force (edit force)" dialogue window. The inputted forces can also be edited on the desktop with the help of active objects.

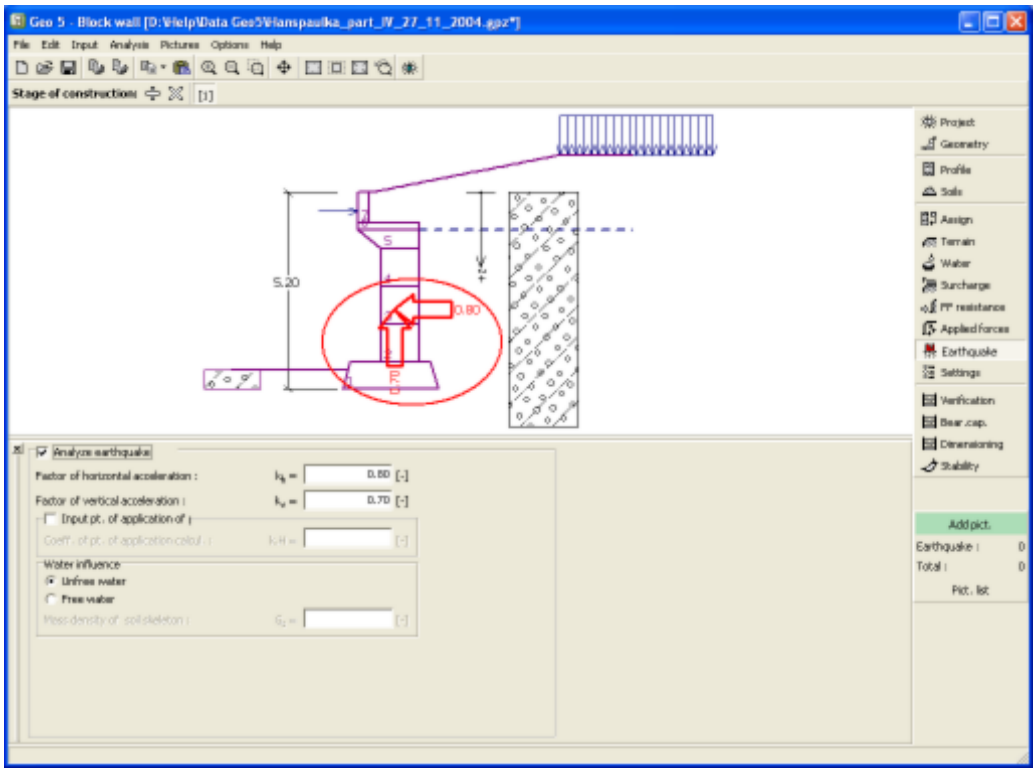


Frame "Inputted forces"

Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of inputted earthquake effects are displayed on the desktop.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the hint in chapter "Influence earthquake".



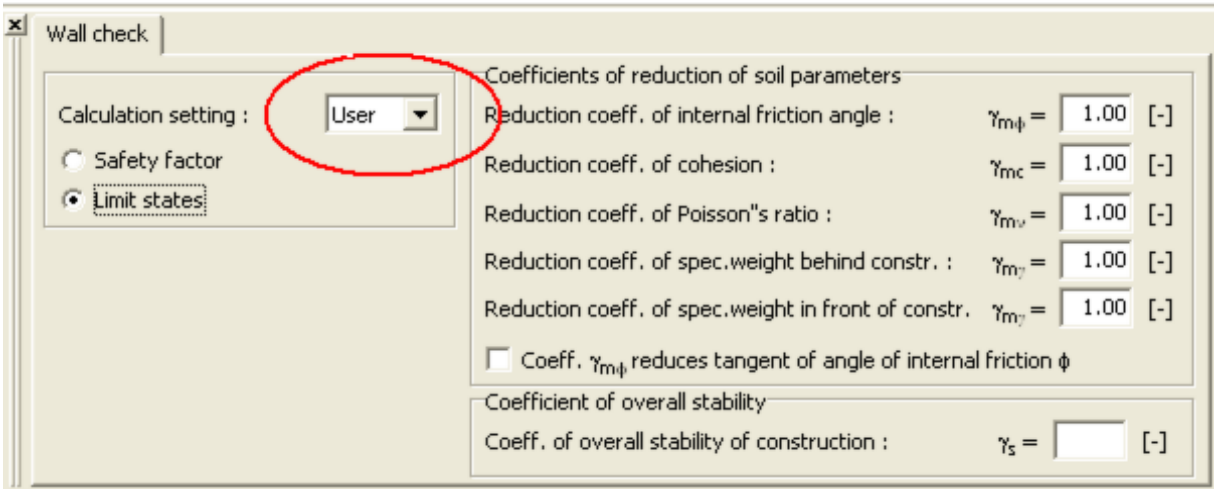
Frame "Earthquake"

Setting

The "Setting" frame contains basic settings for the analysis of earth pressures. The program offers pre-setting for different countries (Czech Republic, Germany, France,...) and " Standard setting" recommended by the authors of the program for countries no included in the list. While changing settings in the combo list the values of coefficients of reduction of soil parameters in corresponding windows are changed.

Evaluating the structure according to theory of limit states also calls for the input of coefficient of overall stability of a structure. When subjecting the wall to overall verification this coefficient is used to multiply the resisting moment M_{res} and the resisting shear force H_{res}.

An arbitrary analysis setting is available with the option "User setting". Selecting the option "Reduce soil parameters" allows for specifying in input fields individual values of the coefficients of reduction of soil parameters (e.g., recommended values according to EC7-1).


Wall check
Calculation setting :
User
Safety factor
Limit states
Coefficients of reduction of soil parameters
Reduction coeff. of internal friction angle : γmφ = 1.00 [-]
Reduction coeff. of cohesion : γmc = 1.00 [-]
Reduction coeff. of Poisson"s ratio : γmν = 1.00 [-]
Reduction coeff. of spec.weight behind constr. : γmγ = 1.00 [-]
Reduction coeff. of spec.weight in front of constr. γmγ = 1.00 [-]
Coeff. γmφ reduces tangent of angle of internal friction φ
Coefficient of overall stability
Coeff. of overall stability of construction : γs = [-]

Frame "Setting"

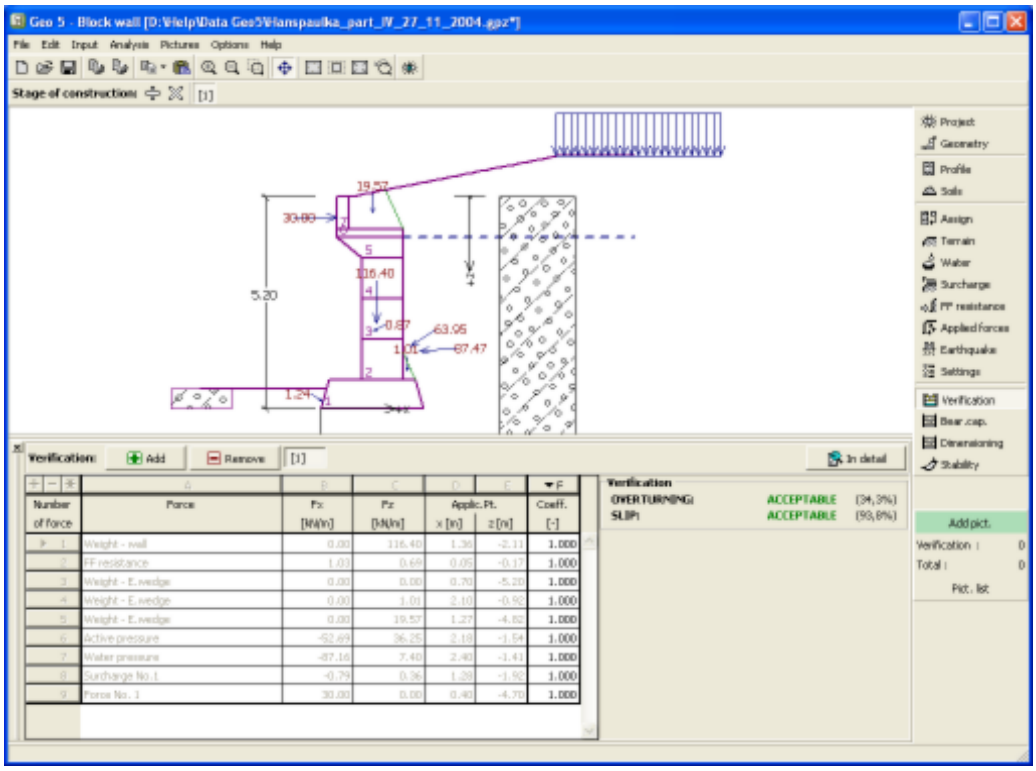
Verification

The "Verification" frame shows the analysis results. Several computation with different coefficients of resultant force effects can be carried out for a single task.

Procedure for wall verification is described in the theoretical part of the hint.

The computed forced are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against overturning and translation. The "In detail" button opens the dialogue window, which contains detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Verification"

Bearing capacity

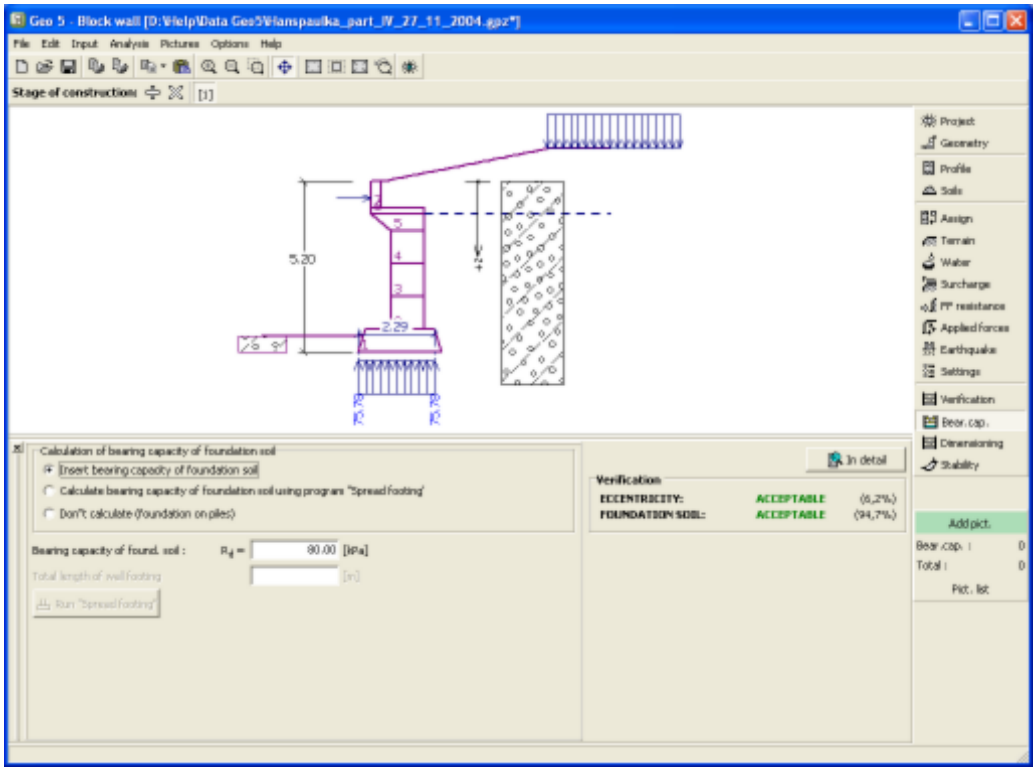
The "Bearing capacity" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all

verifications performed in the frame "Verification". The program "Spread footing" then considers all verifications as loading cases.

Three basic analysis options are available in the frame:

- Input the foundation soil bearing capacity — The input field serves to specify the foundation soil bearing capacity. The results of verification analysis of a soil for eccentricity and bearing capacity are displayed in the right part of the frame. The "In detail" button opens the dialogue window that displays detailed listing of the results of verification analysis of foundation soil bearing capacity.
- Compute the foundation soil bearing capacity using the program "Spread footing" — Pressing the "Run "Spread footing" button starts the program "Spread footing" that allows for computing the soil bearing capacity or settlement and rotation of a footing. Pressing the "OK" button leaves the analysis regime – the results and all plots are copied to the program "Block wall". The program "Spread footing" must be installed for the button to be active.
- Do not compute (pile footing) — The foundation soil bearing capacity is not computed.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Bearing capacity"

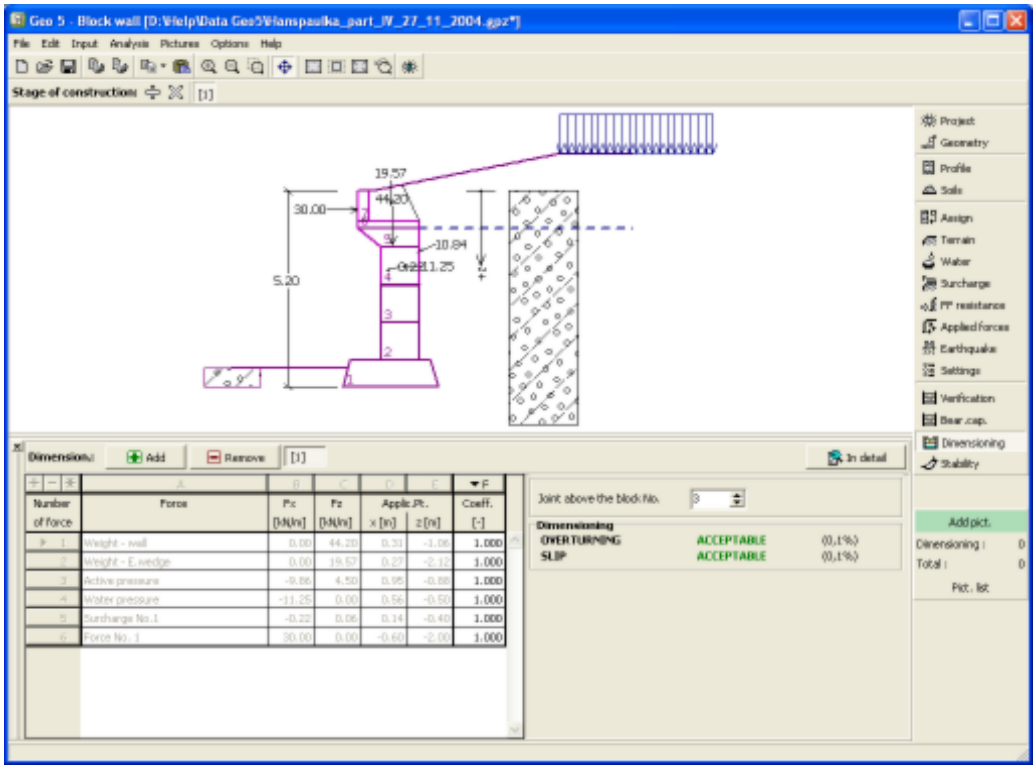
Dimensioning

The "Dimensioning" frame allows for verifying joints between individual blocks of a wall. The "Joint above block No." field serves to select the desired joint subjected to verification analysis. The verification against overturning and translation is performed in the same way as for the entire wall – friction between blocks and cohesion of a block material are inputted in the frame "Geometry".

Several computations for various cross-sections can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the

desktop and are updated with an arbitrary change in data or setting specified in the frame. The "In detail" button opens the dialogue window that contains detailed listing of the dimensioning results.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.

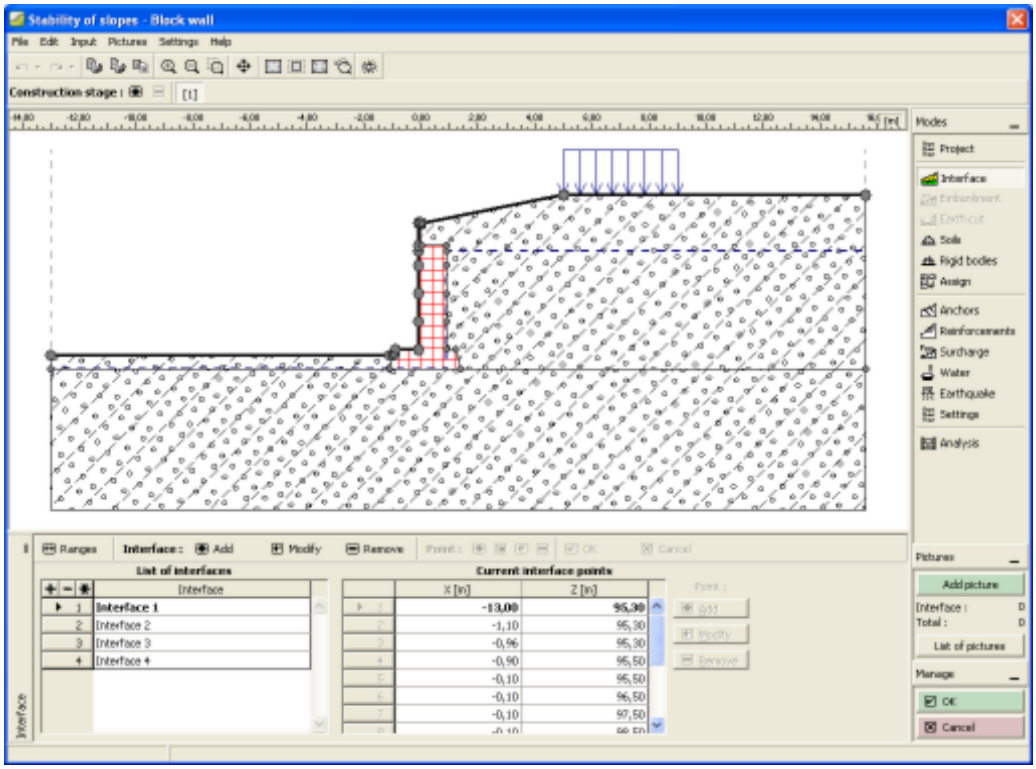


Frame "Dimensioning"

Stability

Pressing the "Stability" button launches the "Slope stability" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "Slope stability" is installed.

After completing all analyses press the "OK" button to leave the program – all data are then carried over to the analysis protocol of the "Block wall" program.



Frame "Stability"

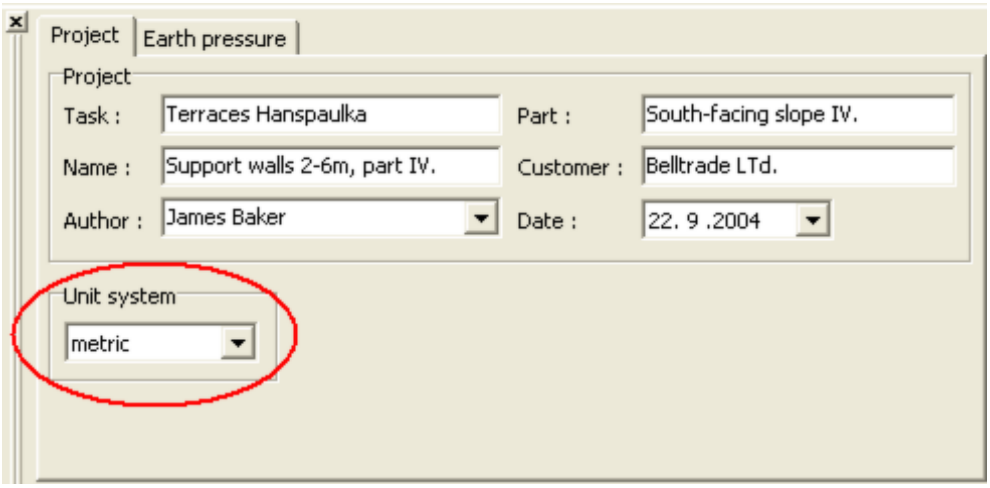
Program Gabion

Project

The "Project" frame is used to input the basic project data and to specify the overall setting of the analysis run. The "Project" tab sheet contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc.

The "Project" tab sheet also allows the user to switch analysis units (metric / imperial).

The "Earth pressures" tab sheet serves to choose the basic theory or standards to be followed in the solution of a given problem.



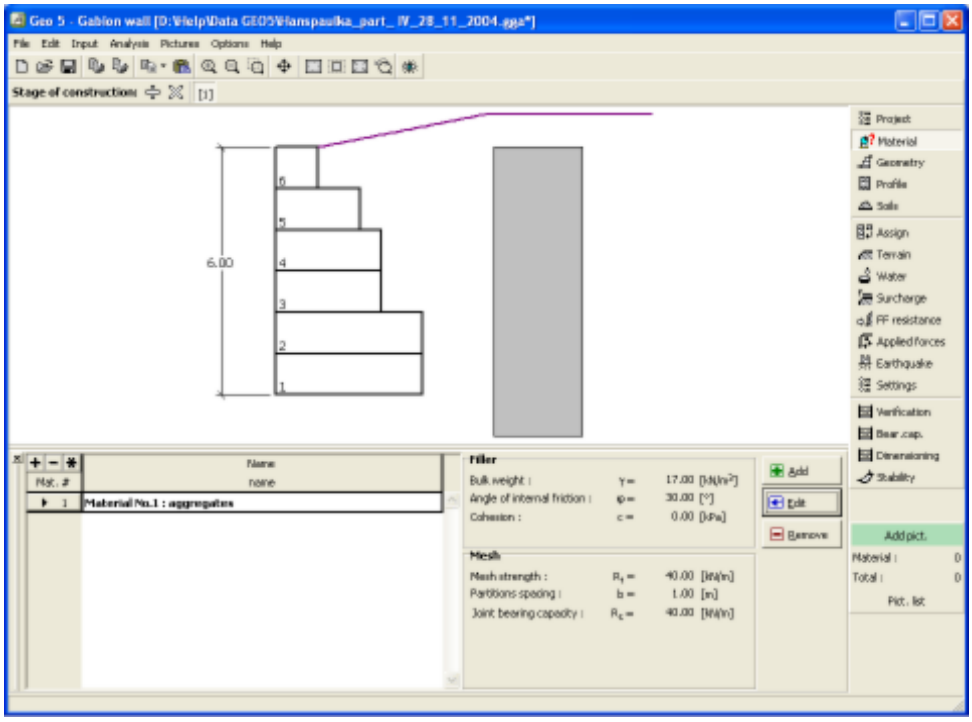
Frame "Project" - tab sheet "Project"

Material

The "Material" frame contains a table with a list of inputted filling (aggregates) and material

parameters of applied gabion wire netting. Adding (Editing) material and netting is performed in the "New material (Edit material)" dialogue window.

The material parameters of filling and netting of currently selected gabion block are displayed in the right part of the frame.



Frame "Material"

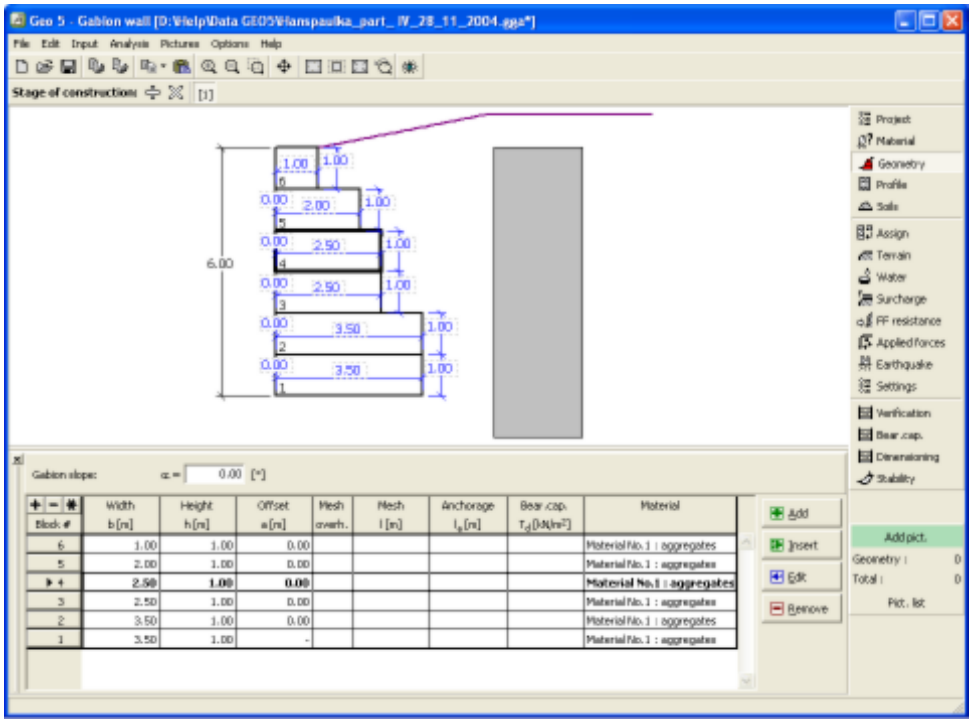
Geometry

The "Geometry" frame contains a table with a list of inputted blocks of a wall (the lowest block is labeled as No. 1). Adding (editing) blocks is performed in the "New block (Edit block)" dialogue window.

This dialogue window serves to define the geometry of a block, and parameters of mesh overhang (overhang length, overhang anchorage, bearing capacity against pull out).

The program allows for adding (inserting) another block in between two already existing blocks of a structure. Inserting a new block is performed in the "Insert block" dialog window that complies with the "New block" dialogue window. The inserted block is ordered such to proceed the currently selected block of a structure.

The inputted blocks can be can be further edited on the desktop with the help of active dimensions or active objects - double clicking on a structure opens a dialog window with a given block. When using the regime of active objects the visualization of detailed dimensions must be turned off in the "Setting visualization style" dialogue window.



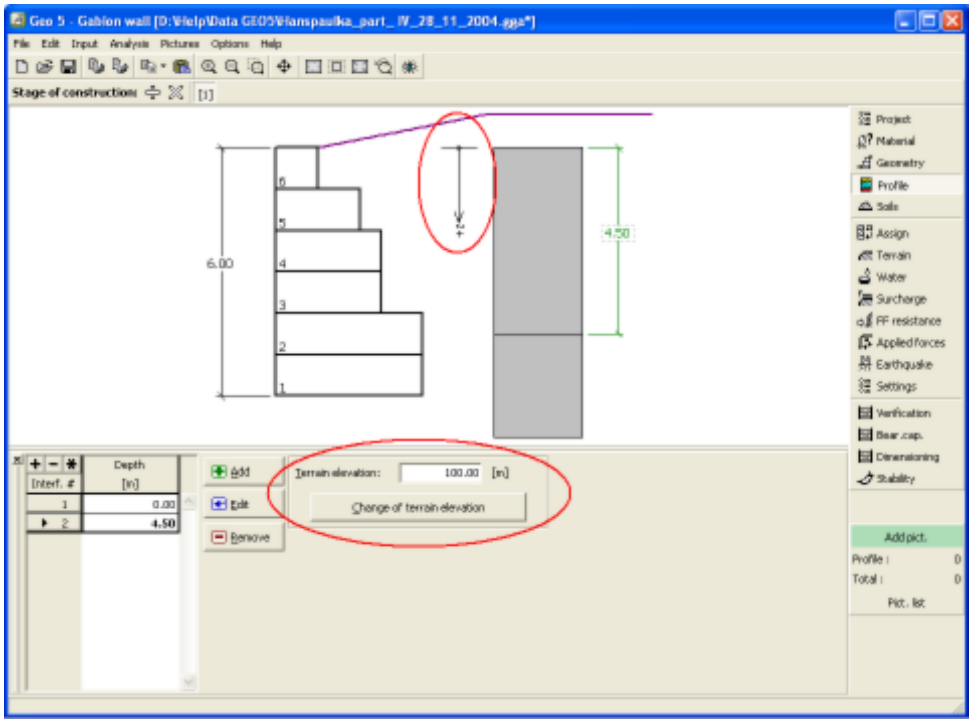
Frame "Geometry"

Profile

The "Profile" frame contains a table with a list of inputted interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "Add (edit) interface" dialogue window. The z-coordinate measured from the top point of a structure is specified.

The program allows for raising or lowering the top point of a structure in the "Change terrain elevation" dialogue window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".



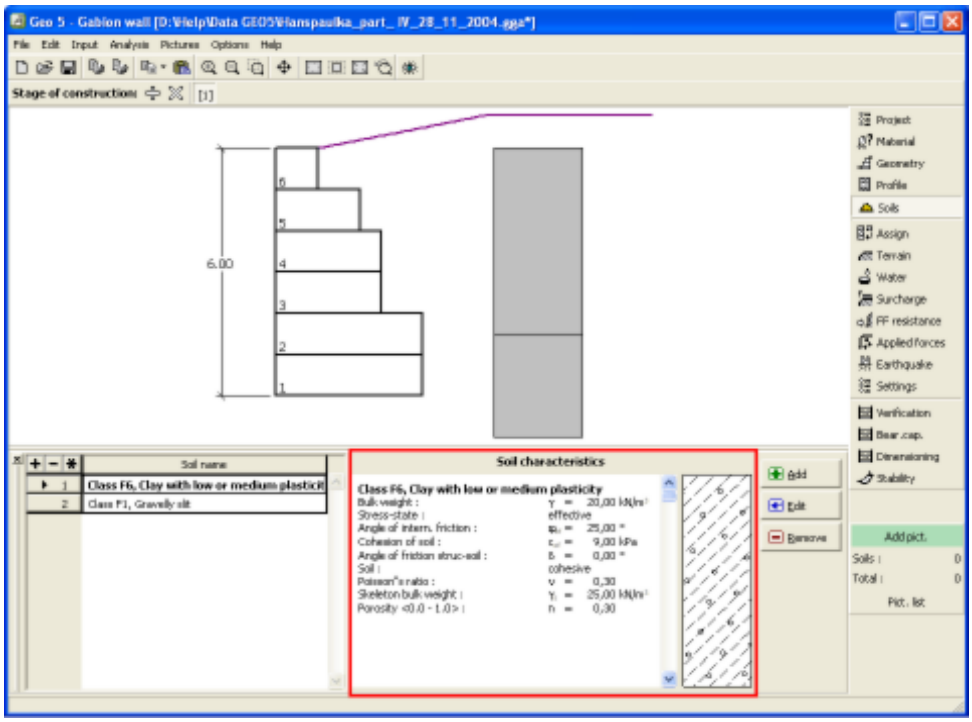
Frame "Profile"

Soils

The "Soils" frame contains a table with a list of inputted soils. The table also provides information about currently selected soil displayed in the right part of the frame.

Adding (editing) a soil is performed in the "Add (edit) soil" dialogue window.

The soil characteristics are specified in the program "Gabion". These characteristics are further specified in chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

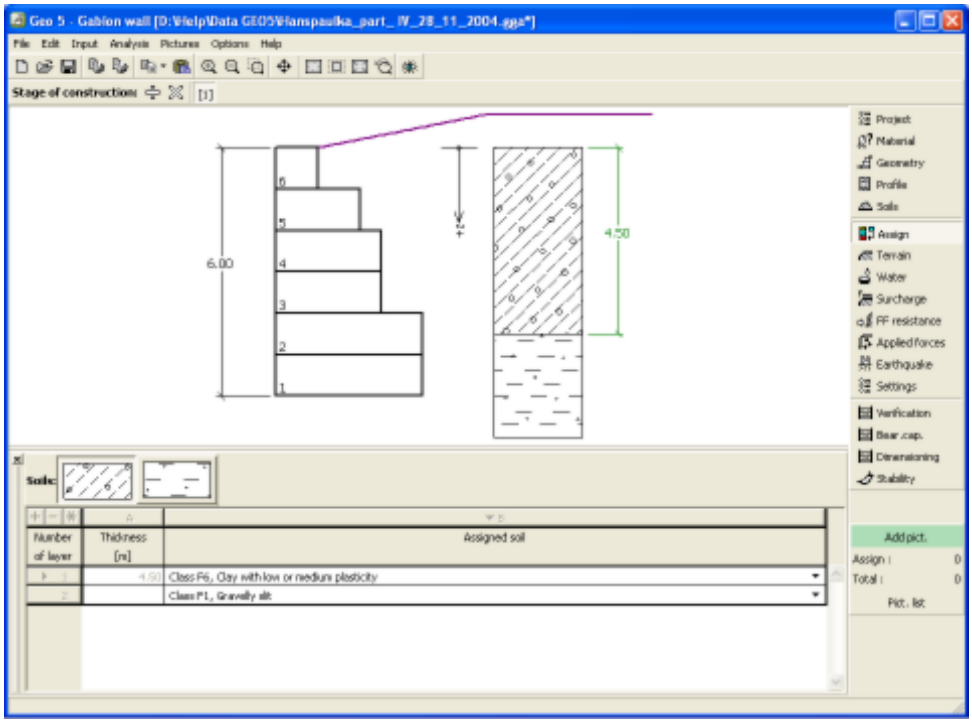


Frame "Soils"

Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

Procedure to assign soil into a layer is described in details herein.



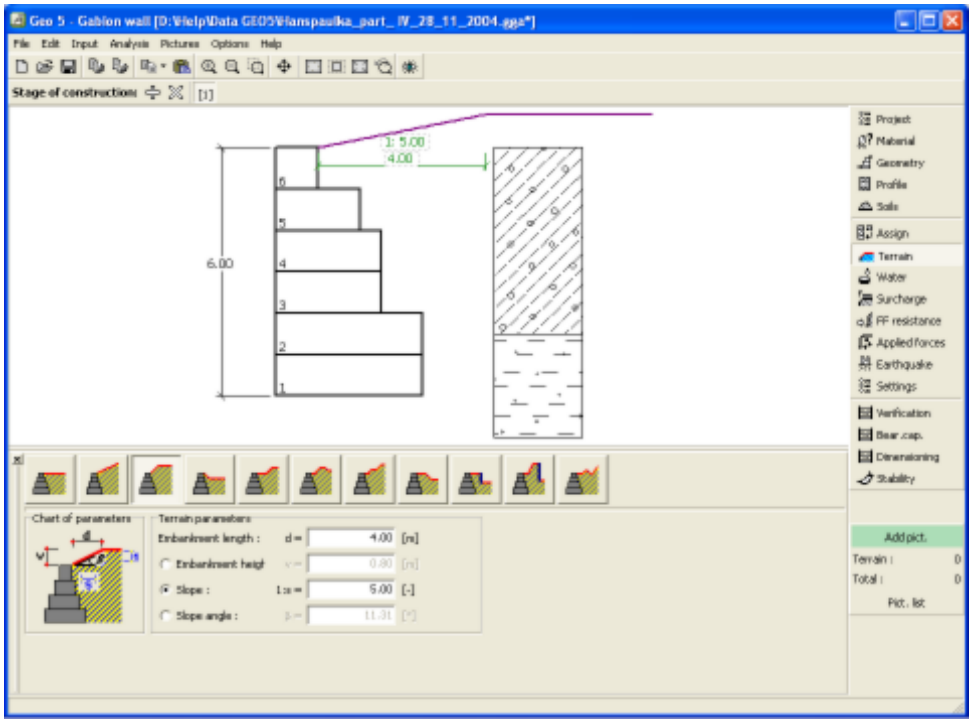
Frame "Assign"

Terrain

The "Terrain" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0, 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the hint, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

The "Water" allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. Water parameters (h1, h2...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

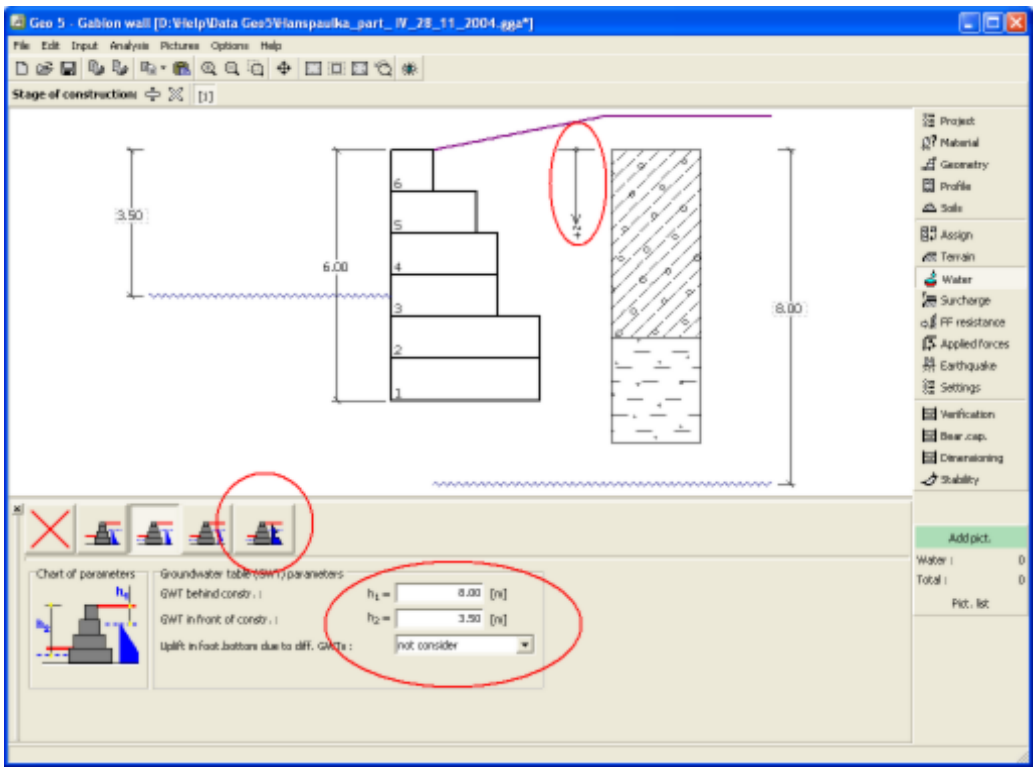
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic or it may not be considered at all. When verifying the wall, the uplift pressure in foundation joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tab sheets "In front of structure" and "Behind structure" appear with tables. The table is filled with values of pore pressure in front, or behind the structure at a depth of "z" (z-axis).

The ground water table can also be specified above the structure or earth profile, respectively – in such a case the depth of water is inputted with negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the hint chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



Frame "Water"

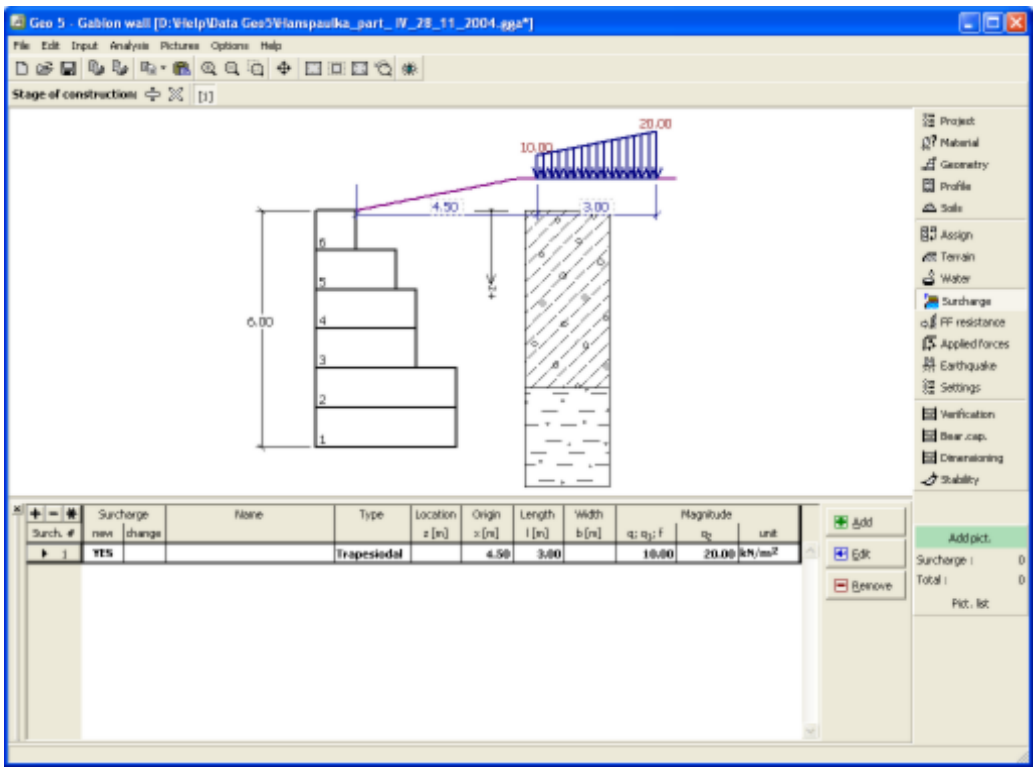
Surcharge

The "Surcharge" frame contains a table with a list of inputted surcharges. Adding (editing) surcharge is performed in the "New (edit) surcharge" dialogue window. The inputted surchages can be edited on the desktop with the help of active dimensions or active objects, respectively.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. Providing the surcharge is found off the terrain the computer prompts an error message.

Analysis of earth pressures due to surcharges is described in the theoretical part of the hint,

chapter "Influence of surcharge".



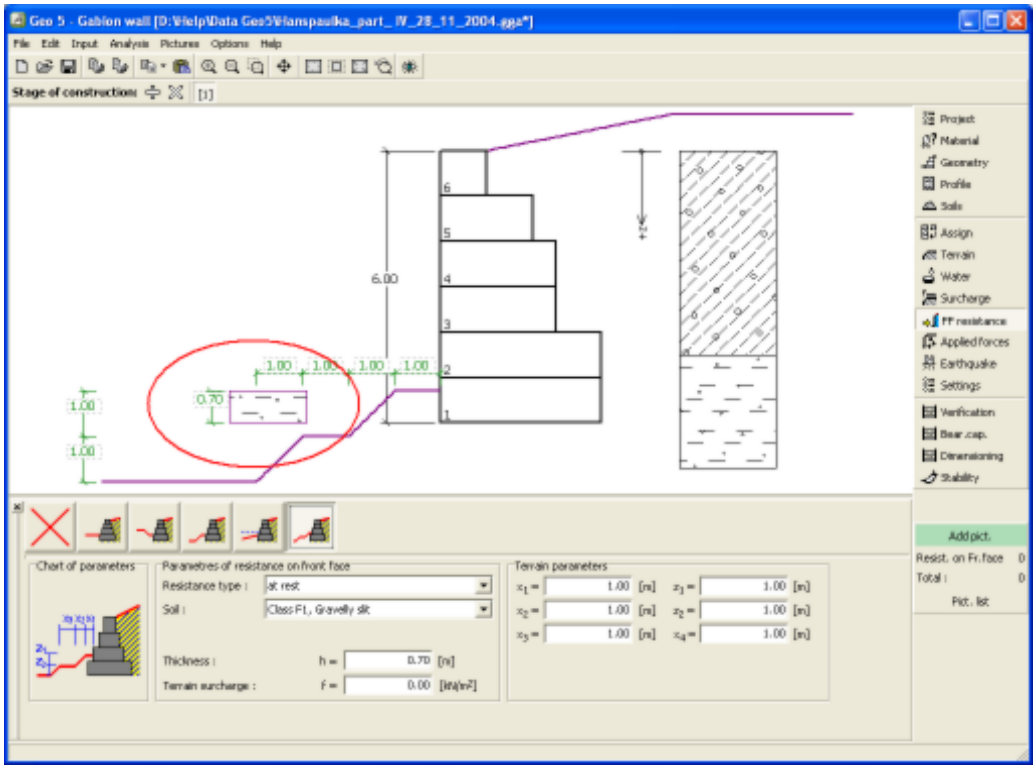
Frame "Surcharge"

Front face resistance

The "Front face resistance" frame allows by pressing the button for specifying theterrain shape and parameters of front face resistance. The selected shape with a graphic hint ("Parameter chart") of inputted values are displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Combo lists in the frame allows the user to select the type of resistance and a soil (the combo list contains soils introduced in the regime "Soils"). The magnitude of terrain surcharge in front of the wall or soil thickness above the wall lowest points can also be specified in the frame.

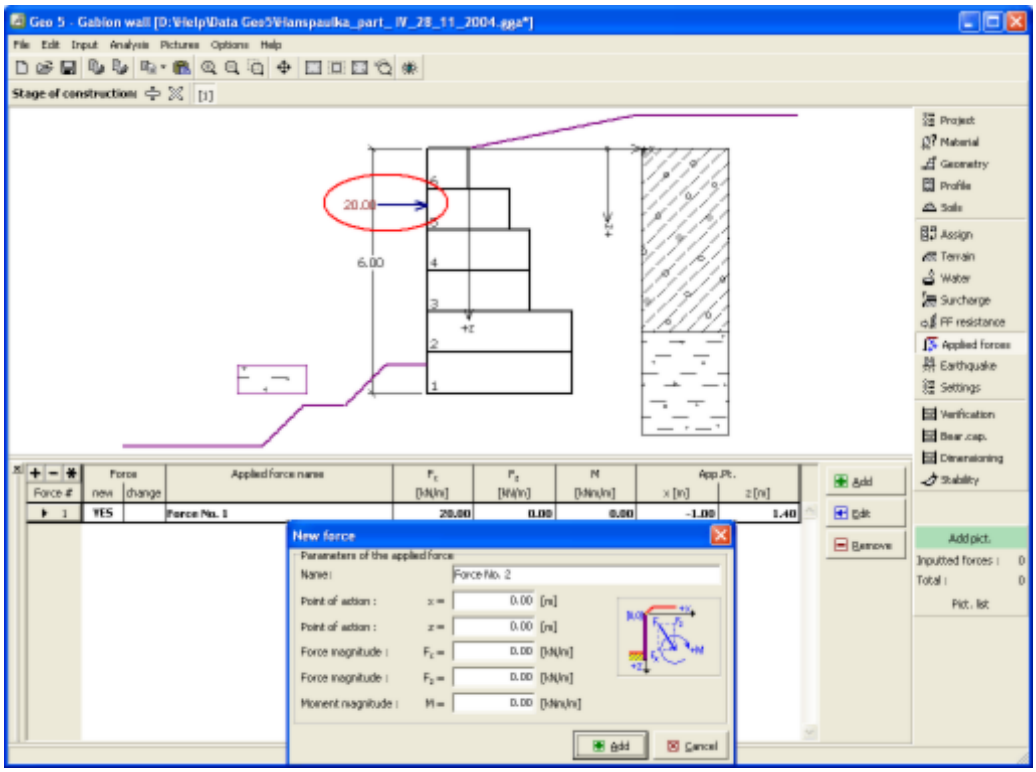
The resistance on a structure front face can be specified as a pressure at rest, passive pressure or reduced passive pressure. The resulting force due to reduced passive pressure is found as a resultant force caused by passive pressure multiplied by a corresponding coefficient, which follows from the inputted type of reduced passive pressure.



Frame "Front face resistance"

Inputted forces

The "Inputted forces" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "New force (edit force)" dialogue window. The inputted forces can also be edited on the desktop with the help of active objects.

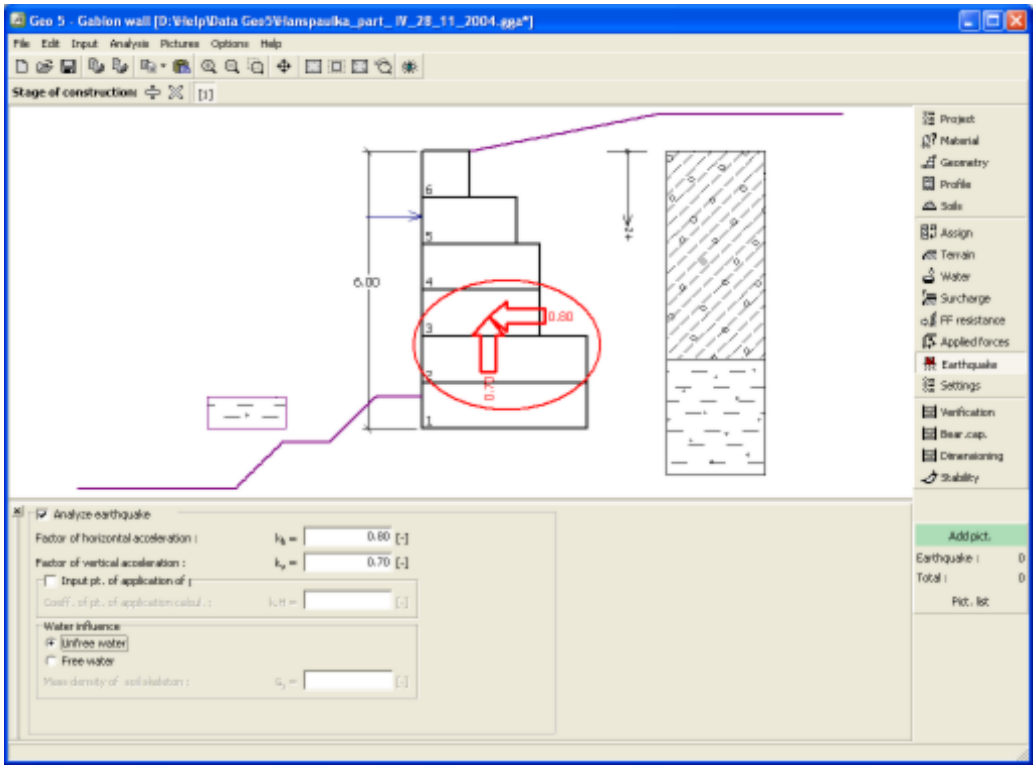


Frame "Inputted forces"

Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of inputted earthquake effects are displayed on the desktop.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the hint in chapter "Influence earthquake".



Frame "Earthquake"

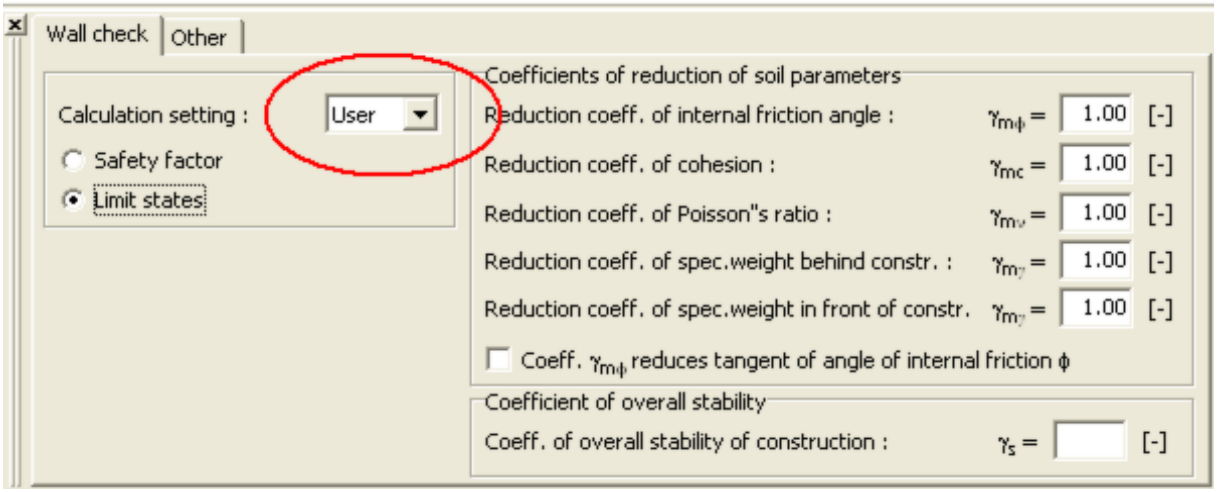
Setting

The "Setting" frame contains basic settings for the analysis of earth pressures. The program offers pre-setting for different countries (Czech Republic, Germany, France,...) and " Standard setting" recommended by the authors of the program for countries no included in the list. While changing settings in the combo list the values of coefficients of reduction of soil parameters in corresponding windows are changed.

Evaluating the structure according to theory of limit states also calls for the input of coefficient of overall stability of a structure. When subjecting the wall to overall verification this coefficient is used to multiply the resisting moment M_{res} and the resisting shear force H_{res}.

Pro An arbitrary analysis setting is available with the option "User setting". Selecting the option "Reduce soil parameters" allows for specifying in input fields individual values of the coefficients of reduction of soil parameters (e.g., recommended values according to EC7).

The "Other" tab sheet allows for specifying the coefficient of reduction of friction between blocks.


Wall check
Other
Calculation setting :
User
Safety factor
Limit states
Coefficients of reduction of soil parameters
Reduction coeff. of internal friction angle : γmφ = 1.00 [-]
Reduction coeff. of cohesion : γmc = 1.00 [-]
Reduction coeff. of Poisson"s ratio : γmν = 1.00 [-]
Reduction coeff. of spec.weight behind constr. : γmγ = 1.00 [-]
Reduction coeff. of spec.weight in front of constr. γmγ = 1.00 [-]
Coeff. γmφ reduces tangent of angle of internal friction φ
Coefficient of overall stability
Coeff. of overall stability of construction : γs = [-]

Frame "Setting"

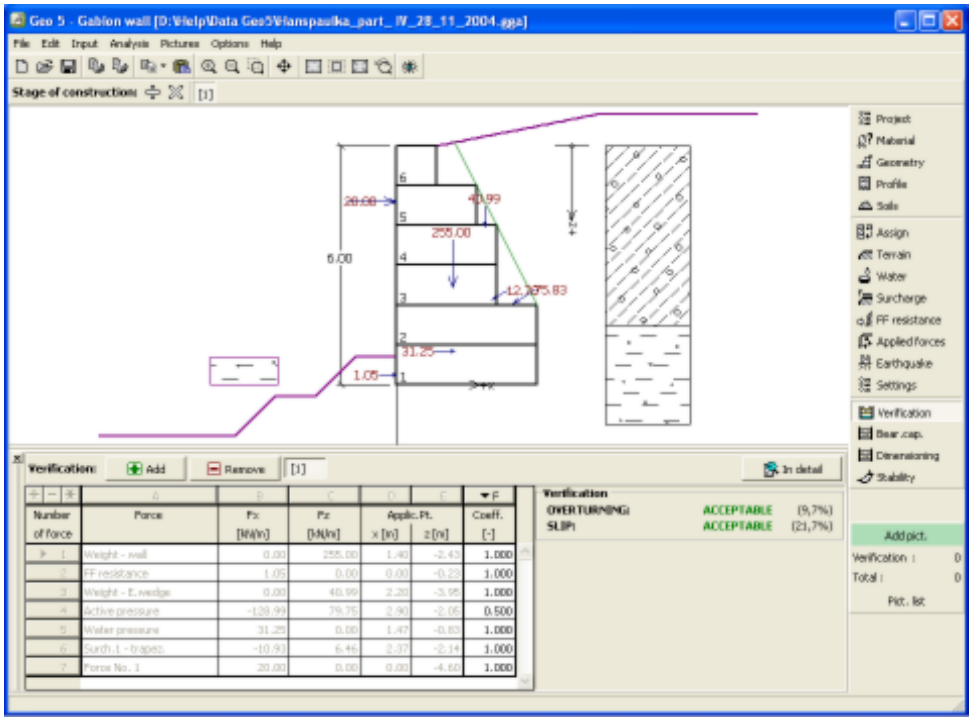
Verification

The "Verification" frame shows the analysis results. Several computation with different design coefficients of resultant force effects can be carried out for a single task.

Procedure for wall verification is described in the theoretical part of the hint.

The computed forced are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against overturning and translation. The "In detail" button opens the dialogue window, which contains detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Verification"

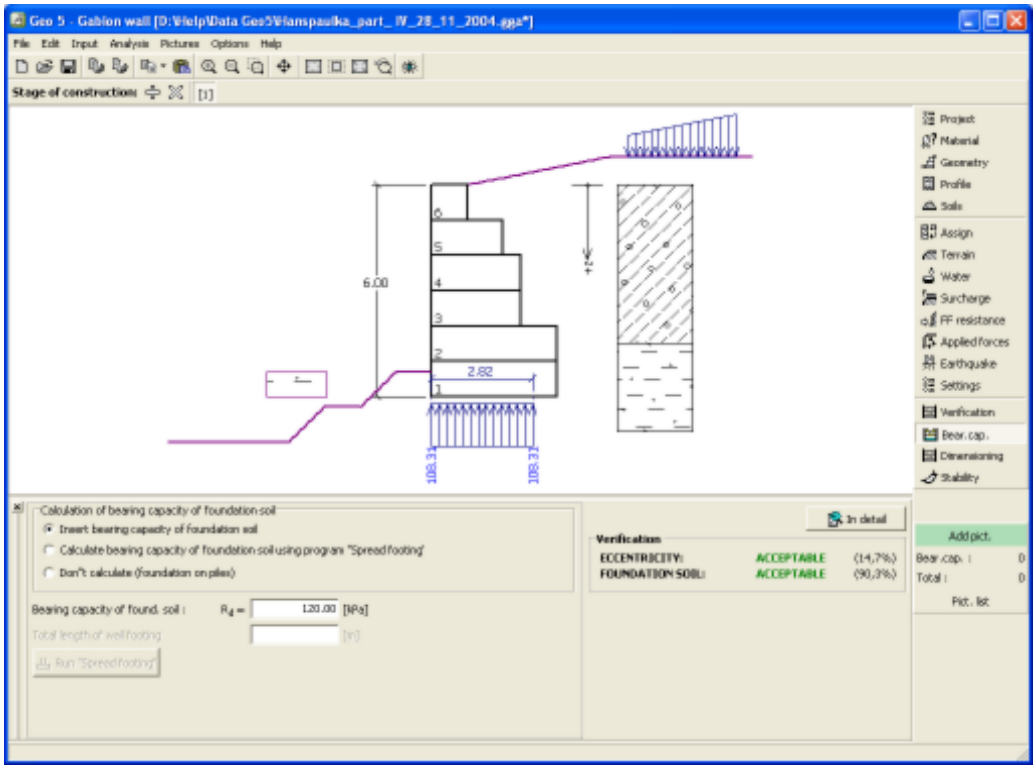
Bearing capacity

The "Bearing capacity" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "Verification". The program "Spread footing" then considers all verifications as loading cases.

Three basic analysis options are available in the frame:

- Input the foundation soil bearing capacity — The input field serves to specify the foundation soil bearing capacity. The results of verification analysis of a soil for eccentricity and bearing capacity are displayed in the right part of the frame. The "In detail" button opens the dialogue window that displays detailed listing of the results of verification analysis of foundation soil bearing capacity.
- Compute the foundation soil bearing capacity using the program "Spread footing" — Pressing the "Run "Spread footing" button starts the program "Spread footing" that allows for computing the soil bearing capacity or settlement and rotation of a footing. Pressing the "OK" button leaves the analysis regime – the results and all plots are copied to the program "Gabion". The program " Spread footing" must be installed for the button to be active.
- Do not compute (pile footing) — The foundation soil bearing capacity is not computed.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



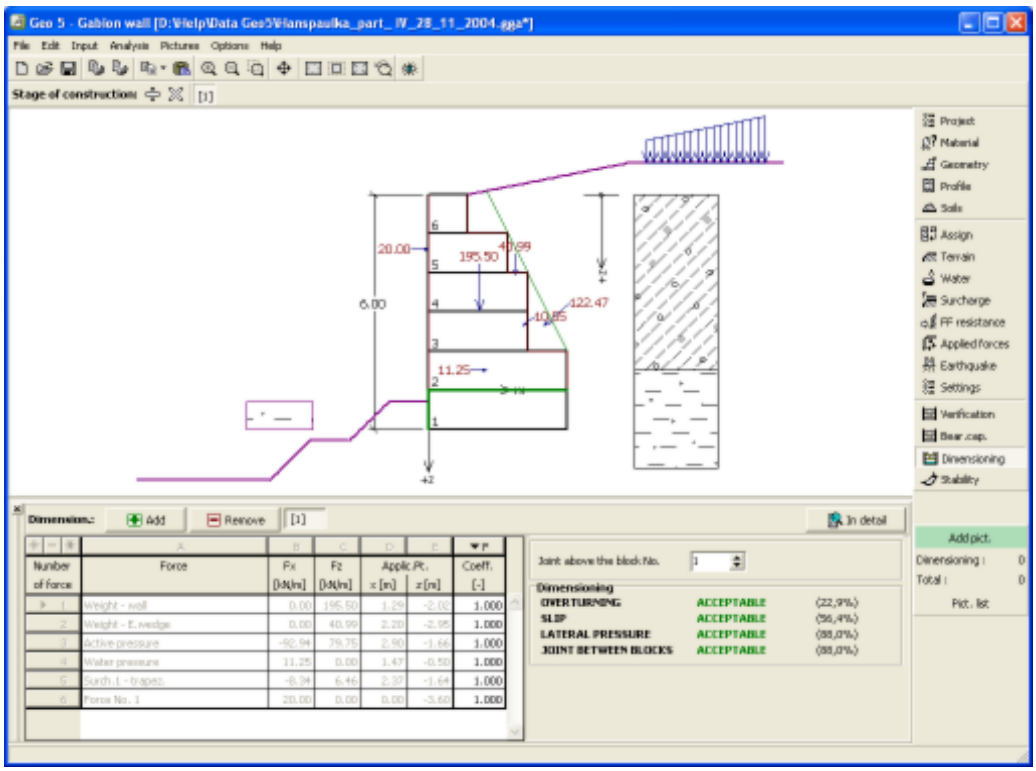
Frame "Bearing capacity"

Dimensioning

The "Dimensioning" frame allows for verifying individual joints of gabion blocks. The "Joint above block No." field serves to select the desired joint subjected to verification analysis. The verification against overturning, translation, for side pressure and joint between blocks is performed.

Several computations for various cross-sections can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "In detail" button opens the dialogue window that contains detailed listing of the dimensioning results.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.

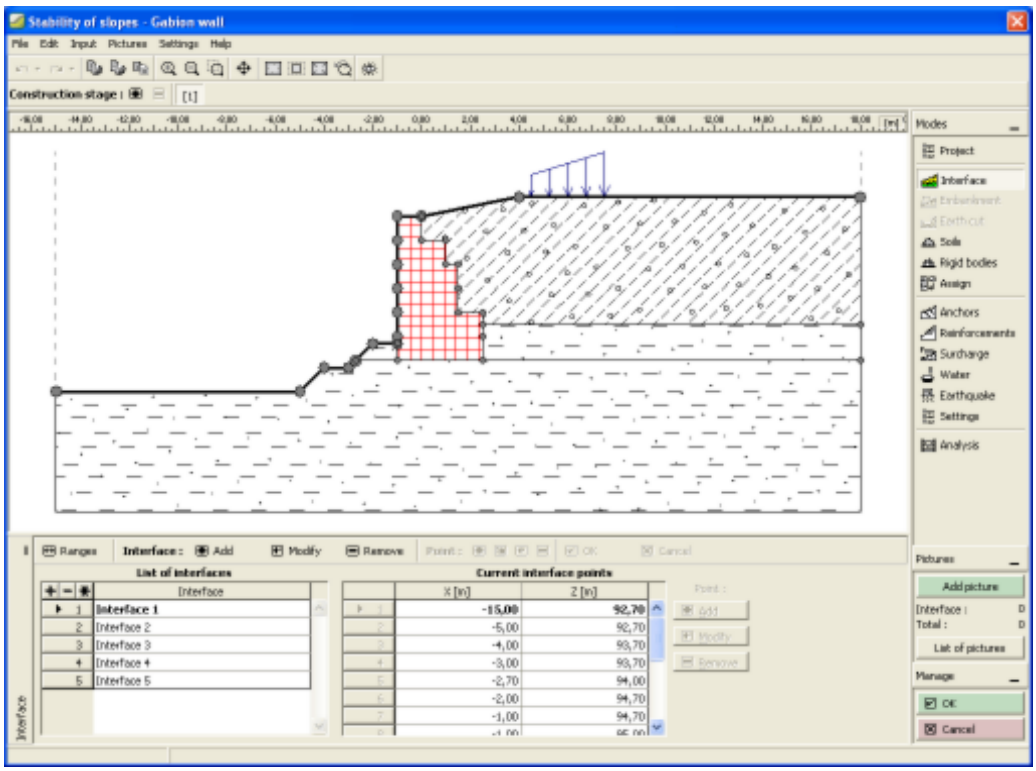


Frame "Dimensioning"

Stability

Pressing the "Stability" button launches the "Slope stability" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "Slope stability" is installed.

After completing all analyses press the "OK" button to leave the program – all data are then carried over to the analysis protocol of the "Gabion" program.



Frame "Stability"

Program Patky

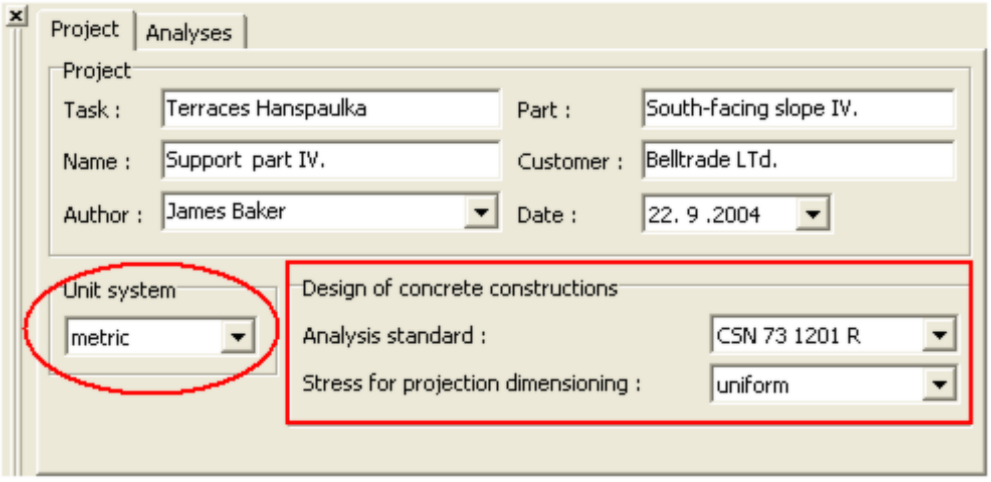
Project

The "Project" frame is used to input the basic project data and to specify the overall setting of the analysis run.

The "Project" tab sheet contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc.

The "Project" tab sheet further serves to specify a standard fornormy se concrete structures dimensioning. Referring to the selected standard the types of concrete and steel are then inputted in the "Material" frame. The dimensioning of cross-sections of the analyzed structure is then performed in the "Dimensioning" frame. Only ACI standard is available when selecting imperial units. While selecting standard it is also necessary to specify whether the stress in the foundation joint used for verification of a wall front key is assumed uniform (ČSN) or trapezoidal (Eurocode).

The "Analysis" tab sheet serves to choose the basic theory or standards to be followed in the solution of a given problem.



Frame "Project" - tab sheet "Project"

Project - Analyses

The "Project" frame – "Analysis" tab sheet contains basic analysis settings. Three variants are available for the analysis of vertical bearing capacity of shallow foundation – foundation on drained subsoil, foundation on undrained subsoil and foundation on rock subsoil.

The "Analysis method" tab sheet then serves to choose the analysis standard or solution procedure, respectively, based on the selected variant.

Options to determine the foundation vertical bearing capacity:

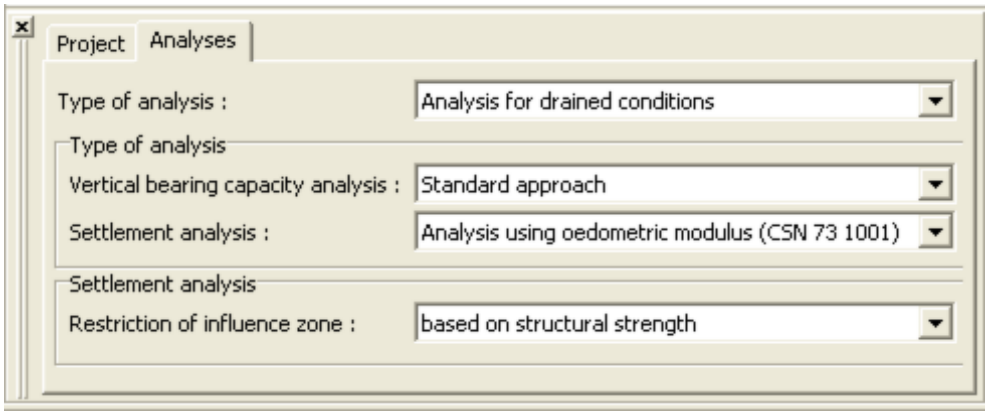
Analysis for drained conditions	Analysis for undrained conditions	Foudation on rock subsoil analysis
• Standard approach	• Standard approach	• Standardní approach
• CSN 73 1001	• CSN 73 1001	• CSN 73 1001
• PN	• PN	• EC7
• IS	• IS	
• EC7	• EC7	

Options for settlement and foundation rotation analysis

- With the help of oedometric modulus
- With the help of compression constant
- With the help of compession index
- NEN (Buissman, Ladd)
- Soft soil model
- Janbu theory
- Analysis based on DMT

Options for determination of depth of influence zone when computing settlement

- Using structural strength
- Using percentage of geostatic stress



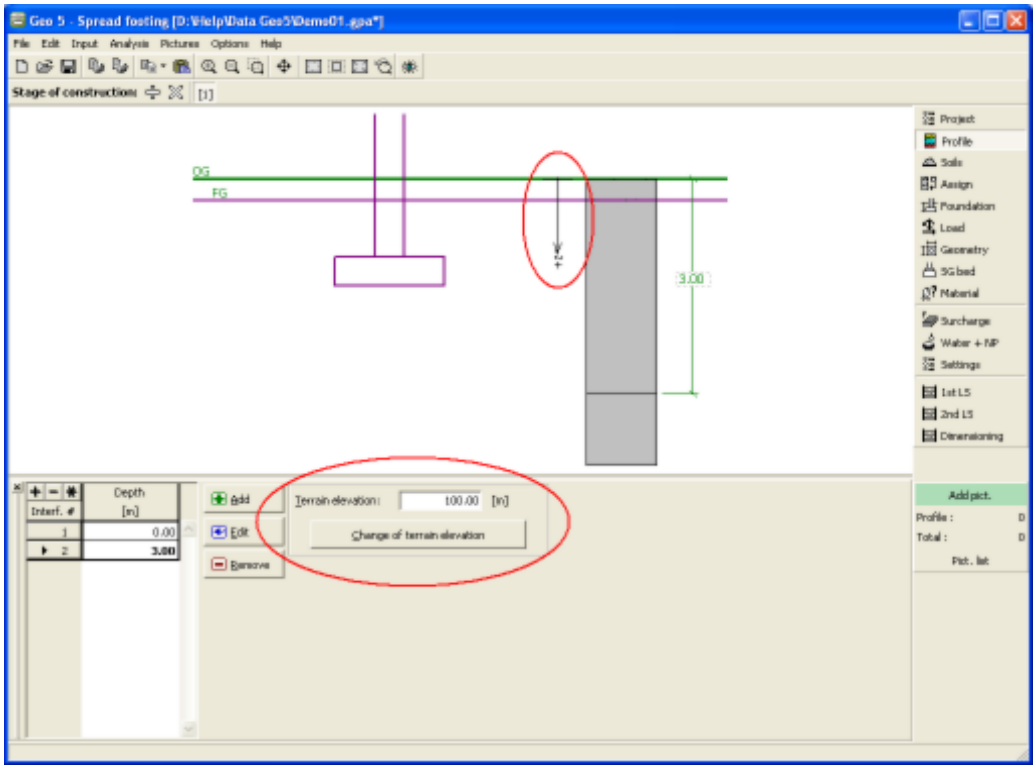
Frame "Project" - tab sheet "Analyses"

Profile

The "Profile" frame contains a table with a list of inputted interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "Add (edit) interface" dialogue window. The z-coordinate measured from the top point of a structure is specified.

The program allows for raising or lowering the top point of a structure in the "Change terrain elevation" dialogue window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".



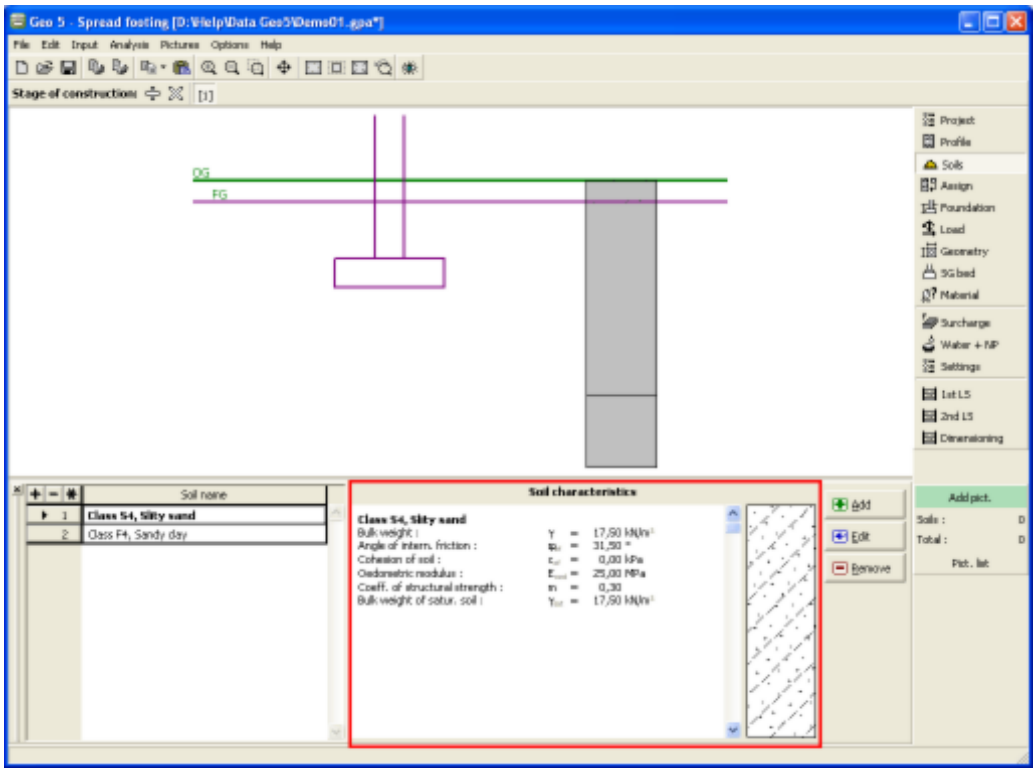
Frame "Profile"

Soils

The "Soils" frame contains a table with a list of inputted soils. The table also provides information about currently selected soil displayed in the right part of the frame.

Adding (editing) a soil is performed in the "Add (edit) soil" dialogue window.

The soil characteristics are specified in the program "Spread footing". These characteristics are further specified in chapters: "Uplift pressure", "Foundation bearing capacity" and "Settlement".

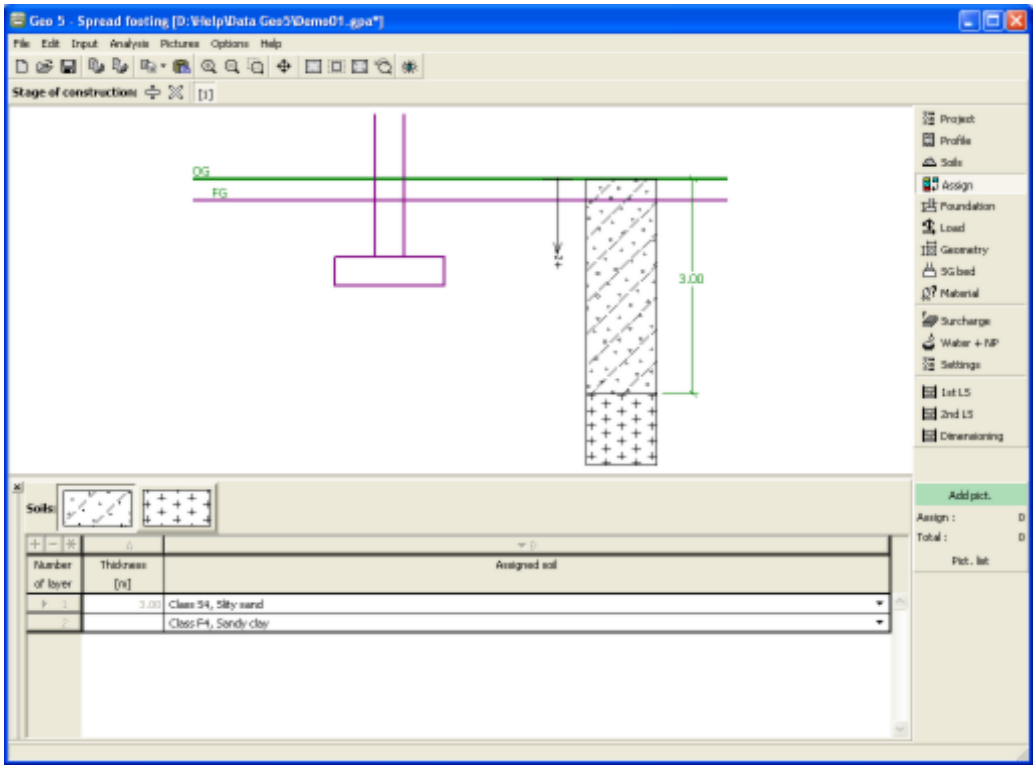


Frame "Soils"

Assign

The “Assign” frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

Procedure to assign soil into a layer is described in details herein.



Frame "Assign"

Foundation

The "Foundation" frame allows for selecting a type of foundation. The selected type with graphic hint ("Geometry chart") of inputted values is displayed in the left part of the frame. The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions. The frame also serves to specify the bulk weight of overburden.

The following types of foundations can be selected:

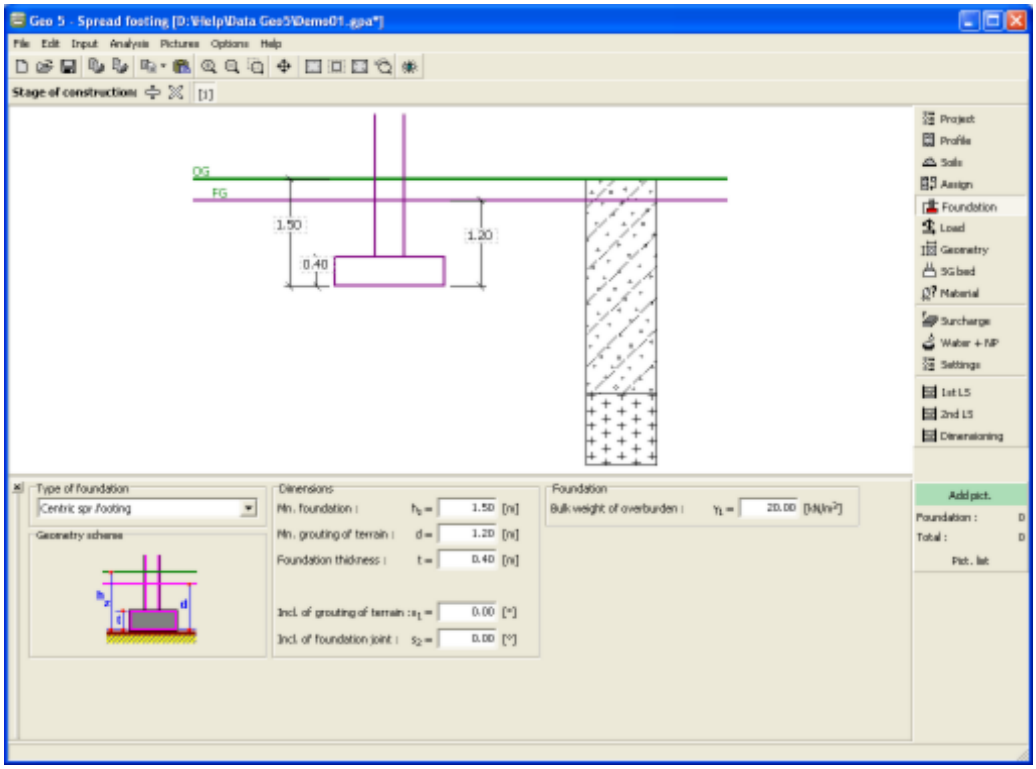
- Centric spread footing
- Eccentric spread footing
- Strip footing
- Stepped centric spread footing
- Stepped eccentric spread footing
- Circular spread footing
- Circular stepped spread
- Centric spread footing with batter
- Eccentric spread footing with batter

The soil profile is specified from the original ground. The foundation bearing capacity depends mainly on the depth of foundation measured from the finished grade. Providing the finished grade is found above the original ground it required to assign the same depth to both the finished grade and original ground and introduce into subsoil a layer with a new made-up-ground. This frame also allows for inputting the foundation thickness.

When completed the foundation is usually filled up with a soil – its bulk weight must be

specified (Overburden bulk weight). Providing the analysis follows the theory of limit states its weight is multiplied by the coefficient inputted in the frame "Setting".

For foundations with an undrained subsoil (type of analysis is selected in the frame "Project" tab sheet "Analysis") it is possible to introduce an inclination of the finished grade and footing bottom. In all other cases both the gound and footing bottom are horizontal.



Frame "Foundation"

Load

The "Load" frame contains a table with a list of inputted loads. Adding (editing) loads is performed in the "New (edit) load" dialogue window. Input of individual forces follows the sign convention displayed in the right part of the dialogue window.

The following types of loading can be specified:

- design load serves to verify the foundation bearing capacity

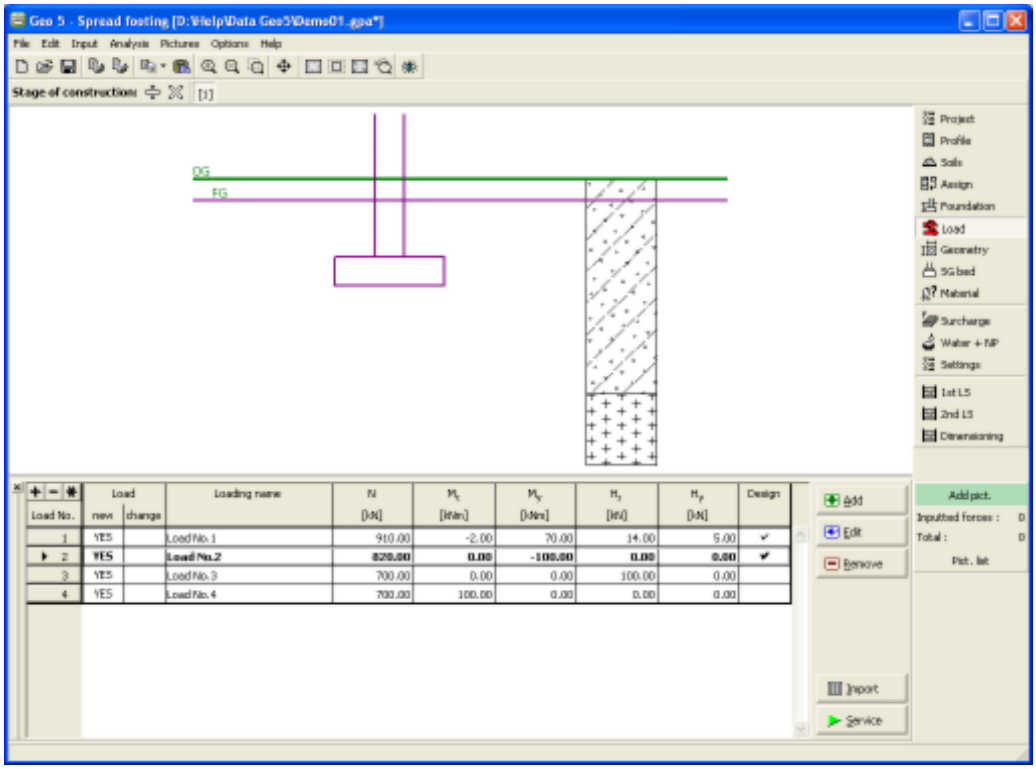
- service load serves to compute the foundation settlement and rotation

Dimensioning of reinforcements assumed for the foundation is carried out for both types of loading.

The foundation is loaded always in the contact point between column and foundation. The program automatically computes the foundation self weight and the weight of overburden.

When inputting loads the program allows by pressing the "Service" button for creating the service loads by dividing the already inputted design loads by design coefficient.

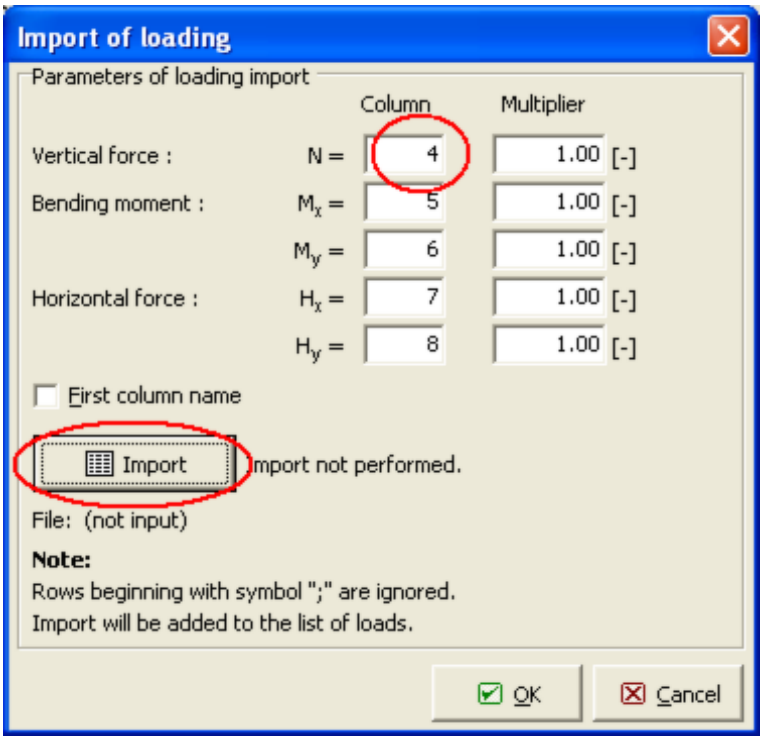
The program also allows for import of loading using the "Import" button.



Frame "Loading"

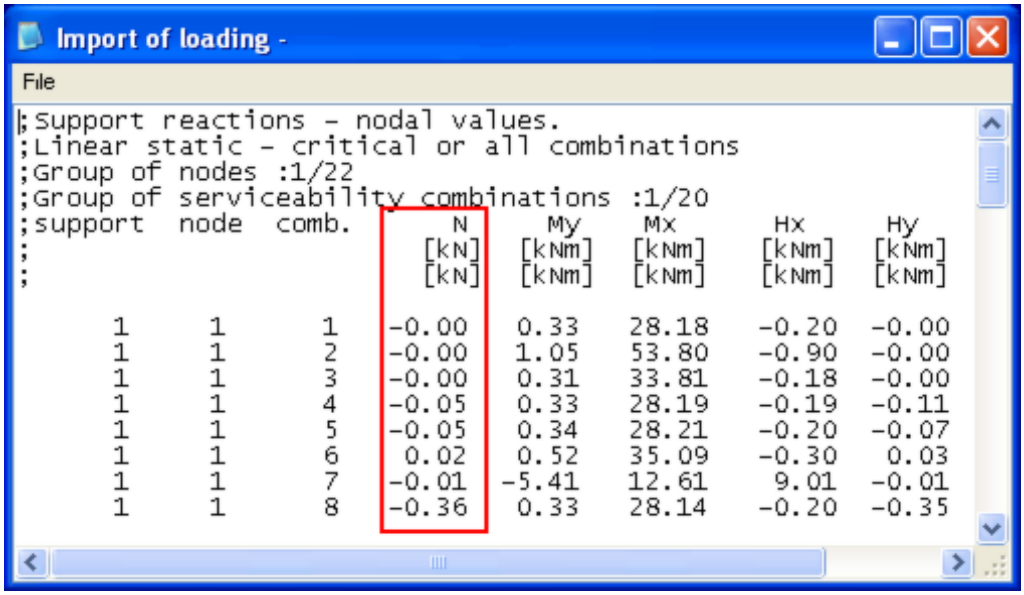
Import of loading

It is possible to import values of loading into the program from text files. Import parameters are set in the "Import loading" dialogue window. In the "Column" input boxes you define the column number in the text file, from which you wish to read data. The "Multiplier" box allows for multiplying the original value by an arbitrary number (e.g. using the number one with a minus sign changes the load direction). The "Import" button opens the "Open" dialogue window, which allows for loading the respective text file. The names of individual loadings if contained by the first column of the text file can loaded while importing the data by checking the "Name in the first column" box.



Dialogue window "Import of loading"

The program allows for importing data from an arbitrary text file (*.TXT). All rows in the text file marked at the beginning by a semicolon are ignored by the program. The figure shows a listing of imported data, where values of individual forces start in the fourth colum n.


Import of loading -
File
;Support reactions - nodal values.
;Linear static - critical or all combinations
;Group of nodes :1/22
;Group of serviceability combinations :1/20
;support node comb. N My Mx Hx Hy
; [kN] [kNm] [kNm] [kNm] [kNm]
; [kN] [kNm] [kNm] [kNm] [kNm]
1 1 1 -0.00 0.33 28.18 -0.20 -0.00
1 1 2 -0.00 1.05 53.80 -0.90 -0.00
1 1 3 -0.00 0.31 33.81 -0.18 -0.00
1 1 4 -0.05 0.33 28.19 -0.19 -0.11
1 1 5 -0.05 0.34 28.21 -0.20 -0.07
1 1 6 0.02 0.52 35.09 -0.30 0.03
1 1 7 -0.01 -5.41 12.61 9.01 -0.01
1 1 8 -0.36 0.33 28.14 -0.20 -0.35

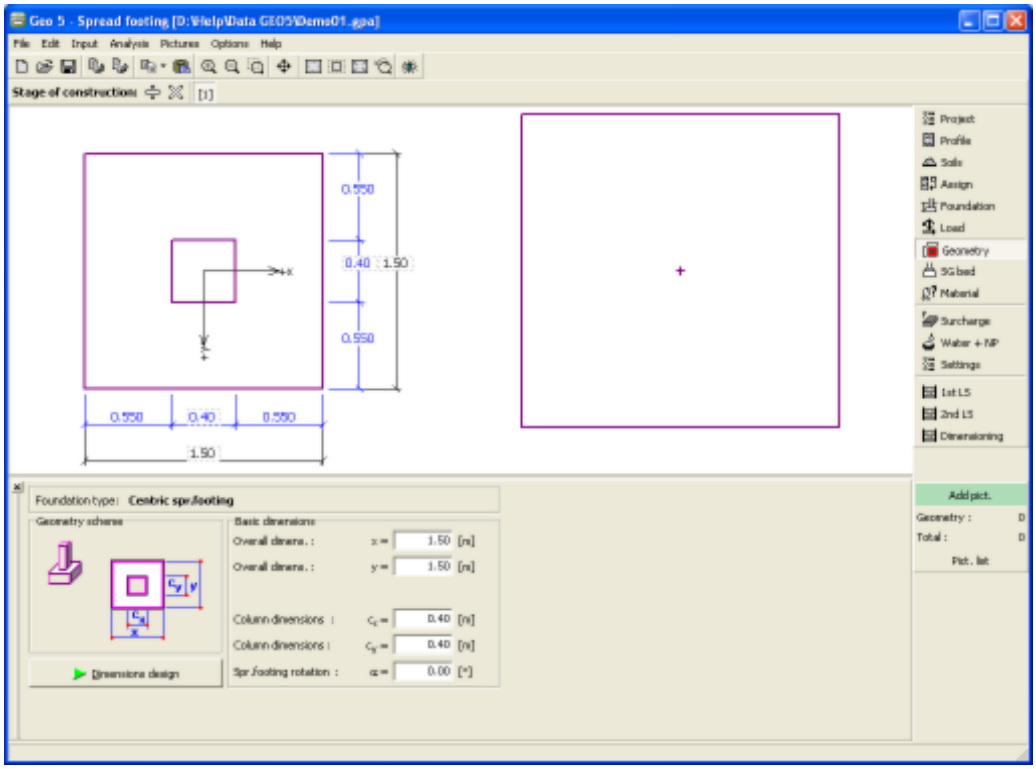
Example of file "Import of loading"

Geometry

The "Geometry" frame allows for specifying the foundation shape. The selected shape with graphic hint ("Geometry chart") of inputted values is displayed in the left part of the frame. The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Foundation type and its thickness are specified in the "Foundation"frame.

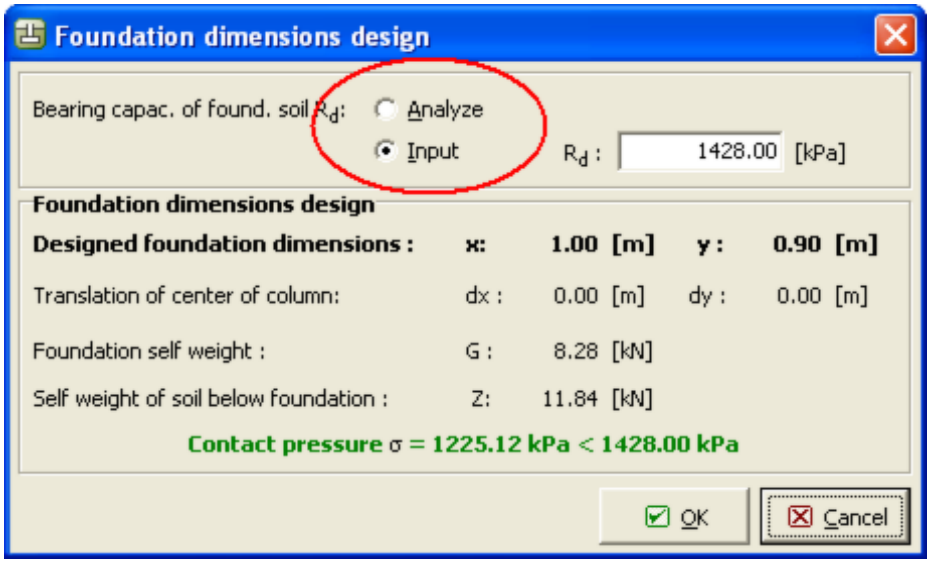
The program automatically computes the self weight of both foundation and overburden above the foundation. The foundation self weight is specified in the "Material" frame. Providing the analysis is carried out employing the theory of limit states the footing self weight is multiplied by coefficients specified in the "Settings" frame.



Frame "Geometry"

The "Dimensions design" button opens the "Foundation dimensions design" that serves with the help of program to compute dimensions of a foundation. The dialogue window allows for inputting the bearing capacity of foundation soil Rd or to select the option "Analyze". In such a case the program determines all dimensions of a foundation based on inputted parameters (soils, profile, water impact, send-gravel-cushion, setting, etc.).

While leaving the dialogue window by pressing the "OK" button the specified dimensions are loaded into the "Geometry" frame.


Foundation dimensions design
Bearing capac. of found. soil Rd: Analyze
Input
Rd : 1428.00 [kPa]
Foundation dimensions design
Designed foundation dimensions : x: 1.00 [m] y : 0.90 [m]
Translation of center of column: dx : 0.00 [m] dy : 0.00 [m]
Foundation self weight : G : 8.28 [kN]
Self weight of soil below foundation : Z: 11.84 [kN]
Contact pressure σ = 1225.12 kPa < 1428.00 kPa
OK
Cancel

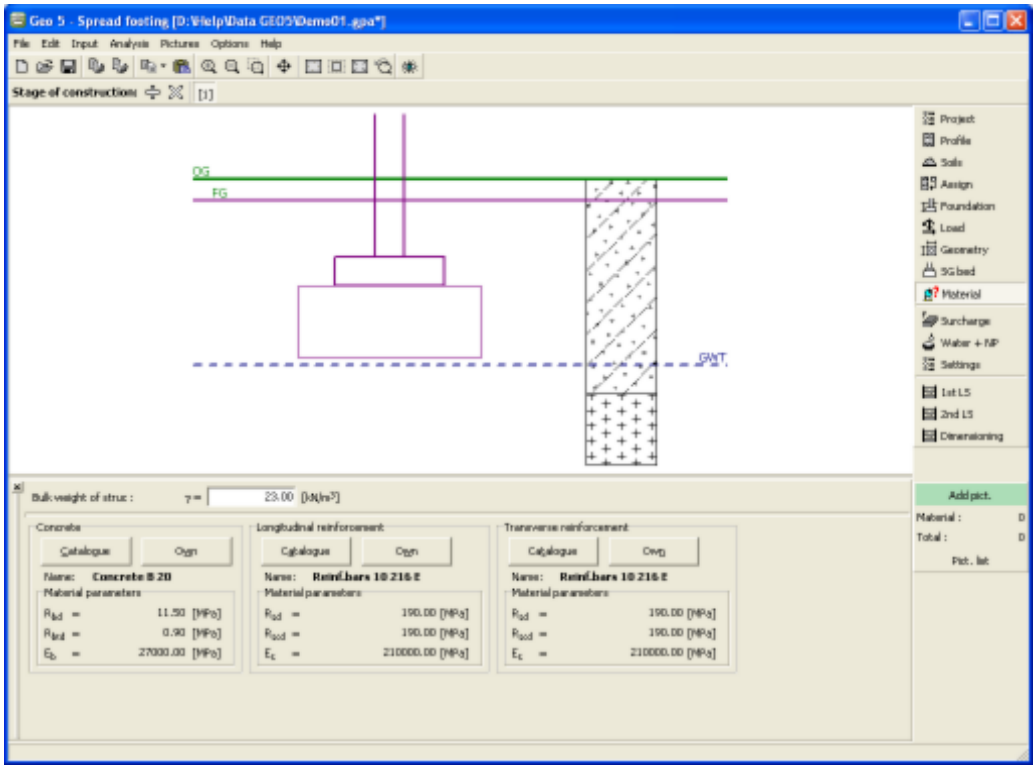
Dialogue window "Foundation dimensions design"

Sand-gravel cushion

The "Sand-gravel cushion" frame allows for inputting parameters of the sand-gravel cushion below foundation. The cushion thickness and overhang over foundation edge are required. The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The cushion filling can be selected from a combo list that contains soils specified in the regime "Soils".

The inputted sand-gravel cushion influences the analysis of both the foundation load bearing capacity and settlement.



Frame "Sand - gravel cushion"

Material

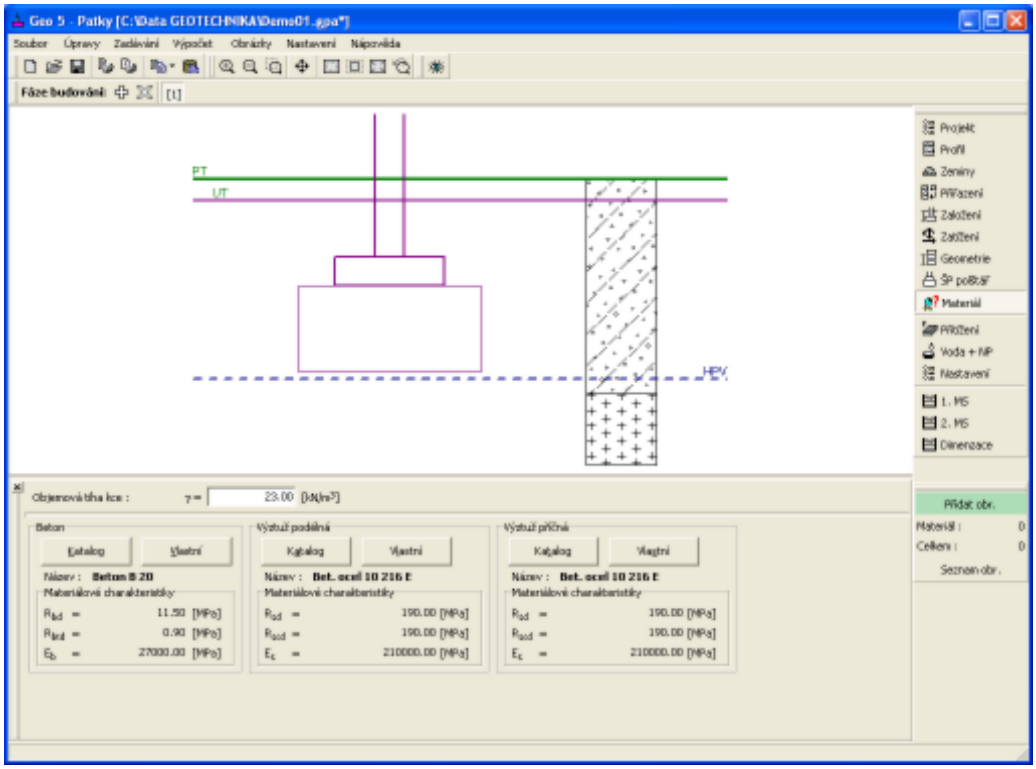
The "Material" frame allows for the selection of material parameters for concrete and longitudinal and transverse steel reinforcements.

Two options are available when selecting the material type:

- the "Catalogue" button opens the "Material catalogue" dialogue window (for concrete or steel reinforcements), the list of materials then serves to select the desired material
- the "User" button opens the "Edit material – concrete" dialogue window (for concrete) or the "Edit material – concrete steel" dialogue window (for longitudinal and transverse steel reinforcements), which allows for manual specification of material parameters

The catalogues content depends on the selection of standard for the design of concrete structures set in the "Project" frame.

The input field in the upper part of the frame serves to specify the wall bulk weight.



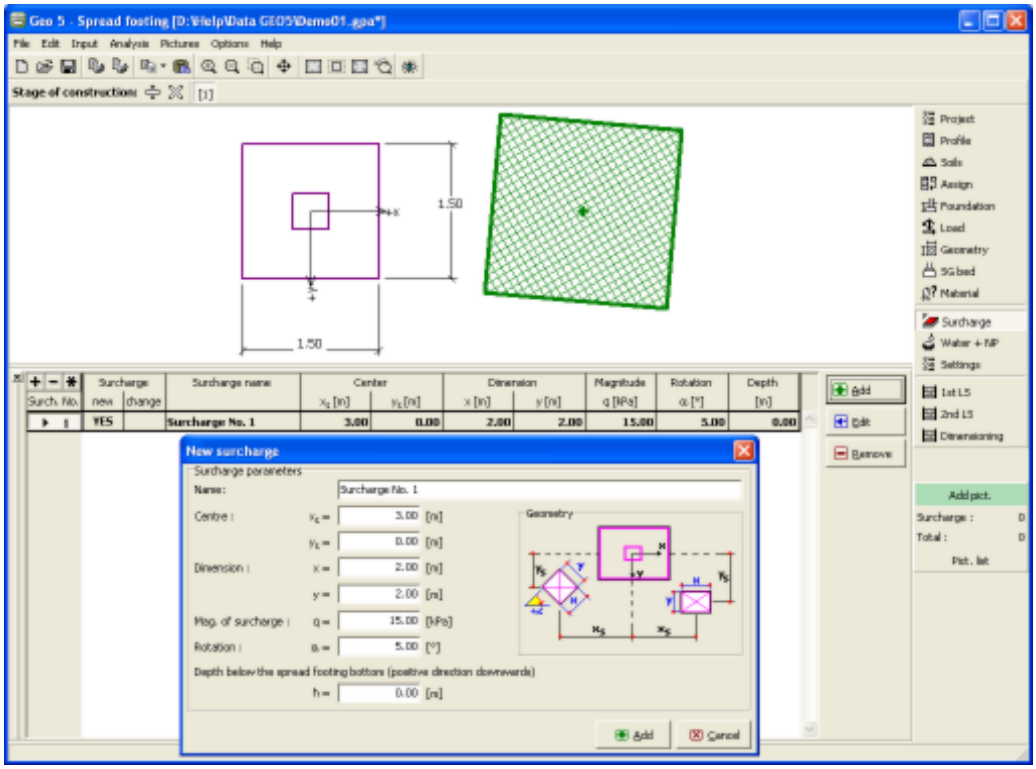
Frame"Material"

Surcharge

The "Surcharge" frame contains a table with a list of inputted surcharges. Adding (editing) surcharge is performed in the "New (edit) surcharge" dialogue window. The values are specified according to "Geometry" chart displayed in the right part of the dialogue window. The inputted surchages can also be edited on the desktop with the help of active objects.

The z-coordinate measured from the foundation joint of a structure is specified (positive direction downwards) when inputting the surcharge at a depth different from the depth of foundation joint.

The surcharge is considered only when computing settlement and rotation of a foundation, in which case it increases the stress in soil below foundation. When computing the foundation bearing capacity, the surcharge is not considered – its presence would increase the bearing capacity.



Frame "Surcharge"

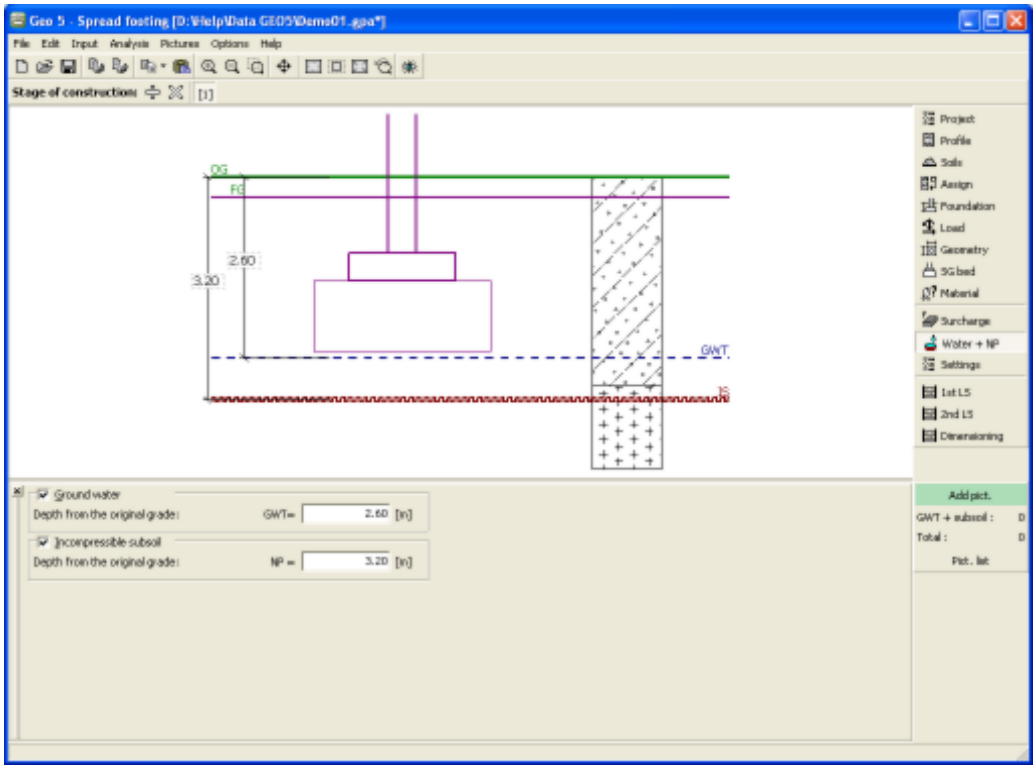
Water, incompressible subsoil

The "Water + IS" frame serves to specify the depth of ground water table and level of incompressible subsoil.

The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The influence of water is manifested by the change of ground water pressure below foundation.

The incompressible subsoil cuts off the influence zone below foundation and also influences reduction in settlement.



Frame "Water, incompressible subsoil"

Setting

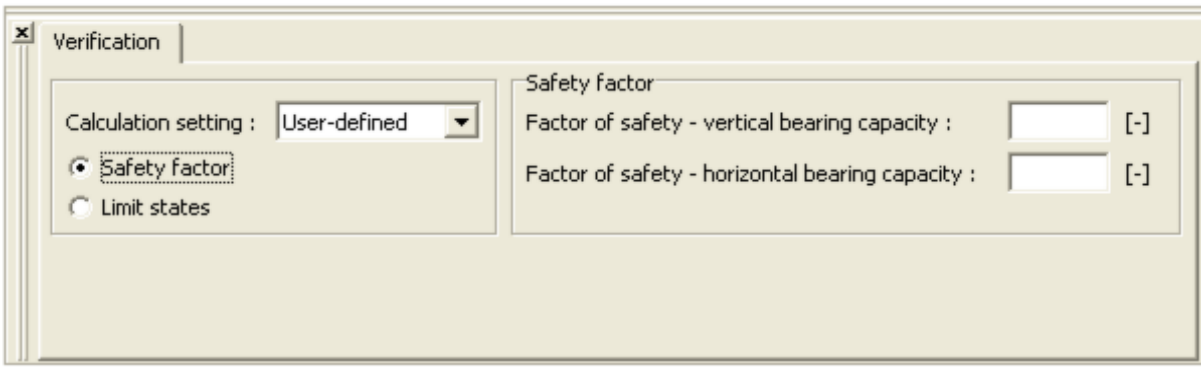
The "Setting" frame contains basic settings to assess the footings. The program offers pre-setting for different countries (Czech Republic, Germany, France,...) and "Standard setting" recommended by the authors of the program for countries no included in the list. While changing settings in the combo list the values of coefficients of reduction of soil parameters in corresponding windows are changed.

An arbitrary analysis setting is available with the option "User setting".

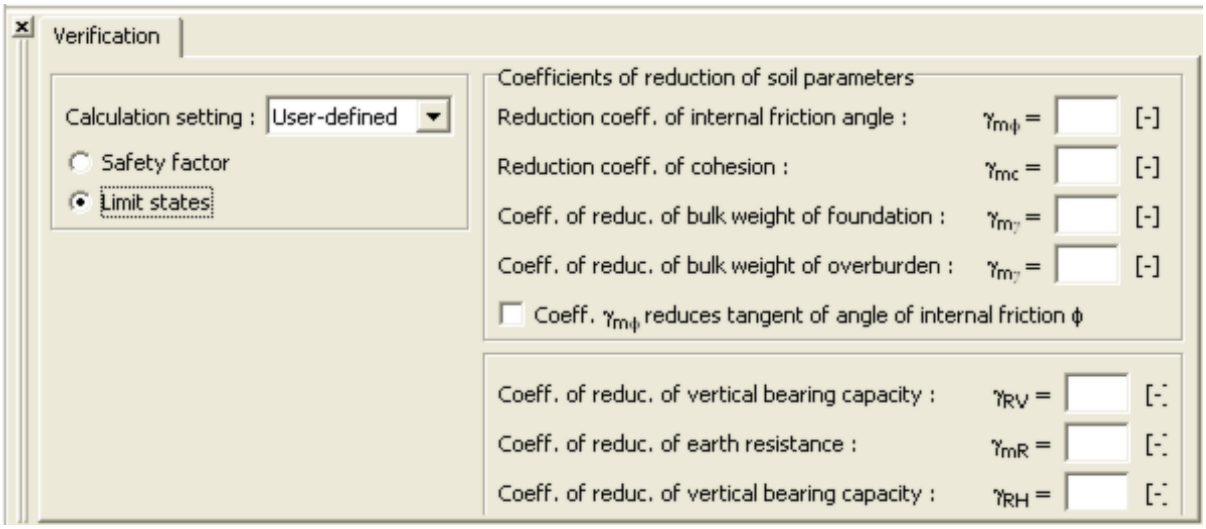
Selecting the option "Safety factor" allows the user to specify in input fields own values of the factor of safety for vertical and horizontal bearing capacity.

Selecting the option "Limit sates" allows for specifying individual values of the coefficients of reduction of soil parameters (e.g., recommended values of coefficients according to EC7) and coefficients for computing the self weight of both foundation and overburden.

The frame further serves for inputting the design coefficients to determine the vertical bearing capacity and the horizontal bearing capacity of a foundation, respectively.



Frame "Setting" - safety factor


Verification
Calculation setting : User-defined
Safety factor
Limit states
Coefficients of reduction of soil parameters
Reduction coeff. of internal friction angle : γmφ = [-]
Reduction coeff. of cohesion : γmc = [-]
Coeff. of reduc. of bulk weight of foundation : γmγ = [-]
Coeff. of reduc. of bulk weight of overburden : γmγ = [-]
Coeff. γmφ reduces tangent of angle of internal friction φ
Coeff. of reduc. of vertical bearing capacity : γRV = [-]
Coeff. of reduc. of earth resistance : γmR = [-]
Coeff. of reduc. of vertical bearing capacity : γRH = [-]

Frame "Setting" - limit states

1.LS - bearing of a footing

The "1. LS" frame serves to verify the verital and horizontal bearing capacity of a footing. More computations can be performed in the frame. The verification can be performed either for individual loads or the program finds the most critical one (can be selected from a combo list).

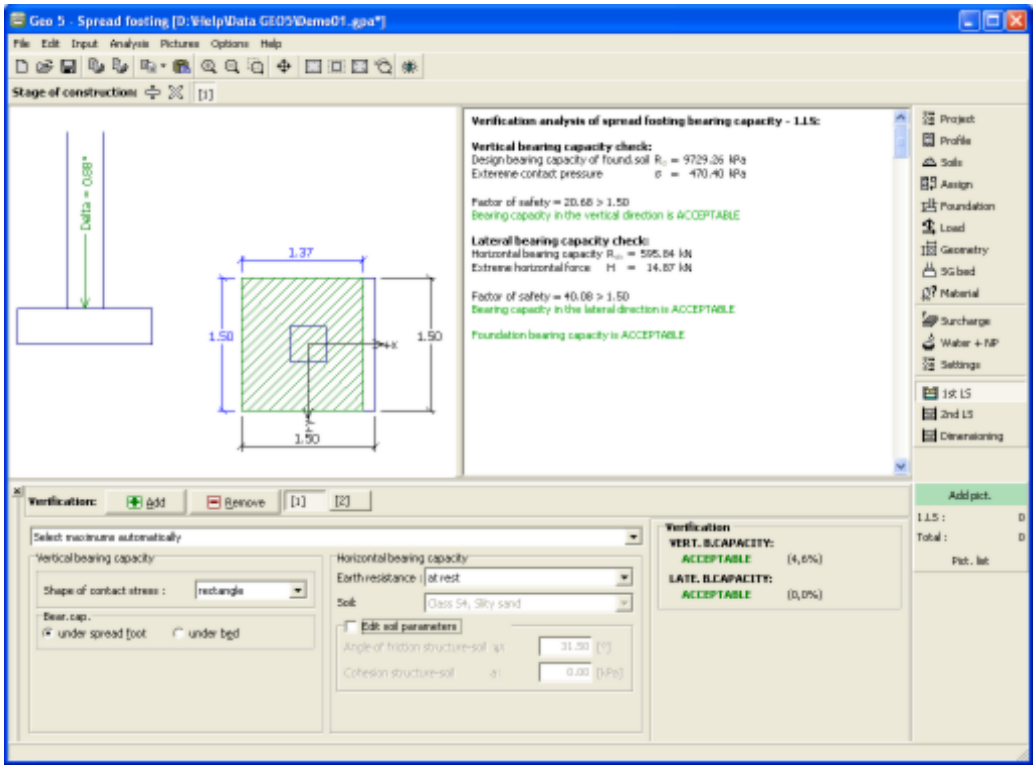
The analysis follows the theory approach selected in the frame "Project" – tab sheet "Analysis". The footing can be verified using either the theory of limit states or the factor of safety concept specified in the frame "Setting".

The vertical bearing capacity analysis requires selection of the type of contact pressure (general shape, rectangle). The shape of contact pressure is plotted in the left part of the desktop.

The horizontal bearing capacity analysis requires selection of the type of earth resistance that can be assumed as the pressure at rest, passive pressure or the reduced passive pressure.

The soil parameters (friction angle structure-soil, cohesion structure-soil) can be further reduced when computing the horizontal bearing capacity.

Detailed listing of the results is displayed in the right part of the desktop. Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "1.LS - bearing of a footing"

2.LS - settlement and rotation of a footing

The "2.LS" frame serves to compute the foundation settlement and rotation. The frame allows for more analyses. The verification can be performed either for individual loads or the program finds the most critical one (can be selected from combo list).

The analysis of foundation settlement and rotation is carried out according to the theory specified in the frame "Project" tab sheet "Analysis".

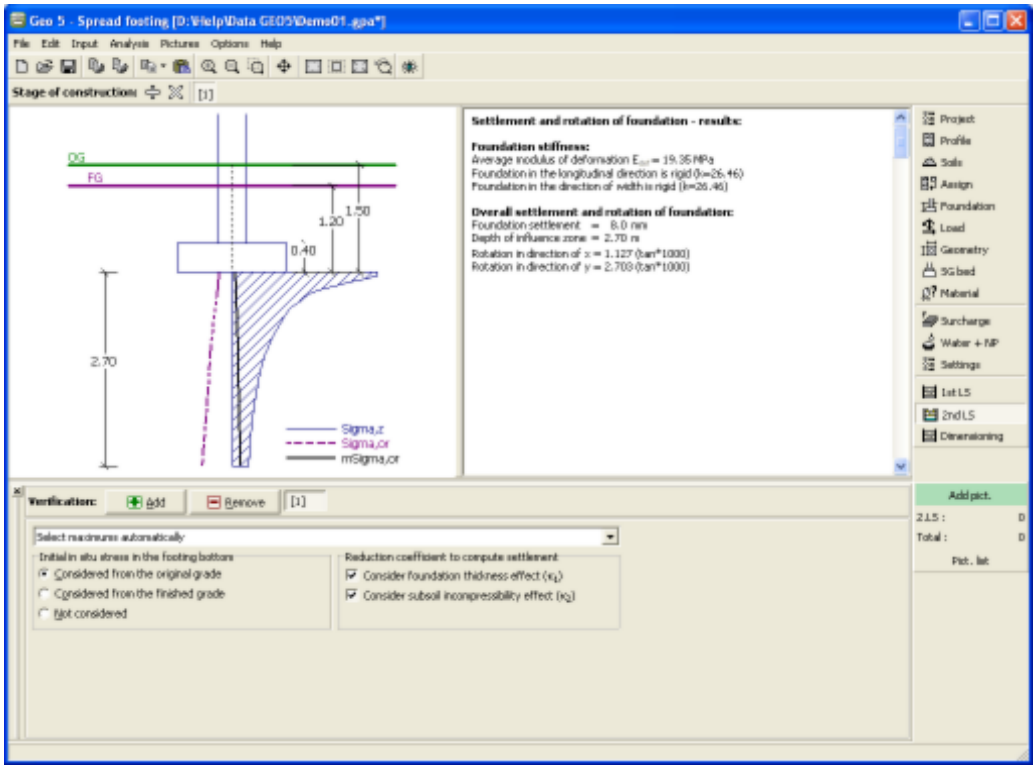
The stress in the footing button can be subtracted from the geostatic stress given by:

- original ground
- finished grade
- not specified

Distributions of the geostatic stress and the stress increment below foundation are displayed in the left part of the desktop. The label below footing represents the depth of deformation zone. The stress is drawn below footing at the point with a characteristic deformation.

The frame also allows for specifying the coefficient of reduction of computation of settlement.

The detailed listing of the verification analysis results is displayed in the right part of the desktop. It can be adjusted in the "Setting visualization style" dialogue window.



Frame "2. LS - settlement and rotation of a footing"

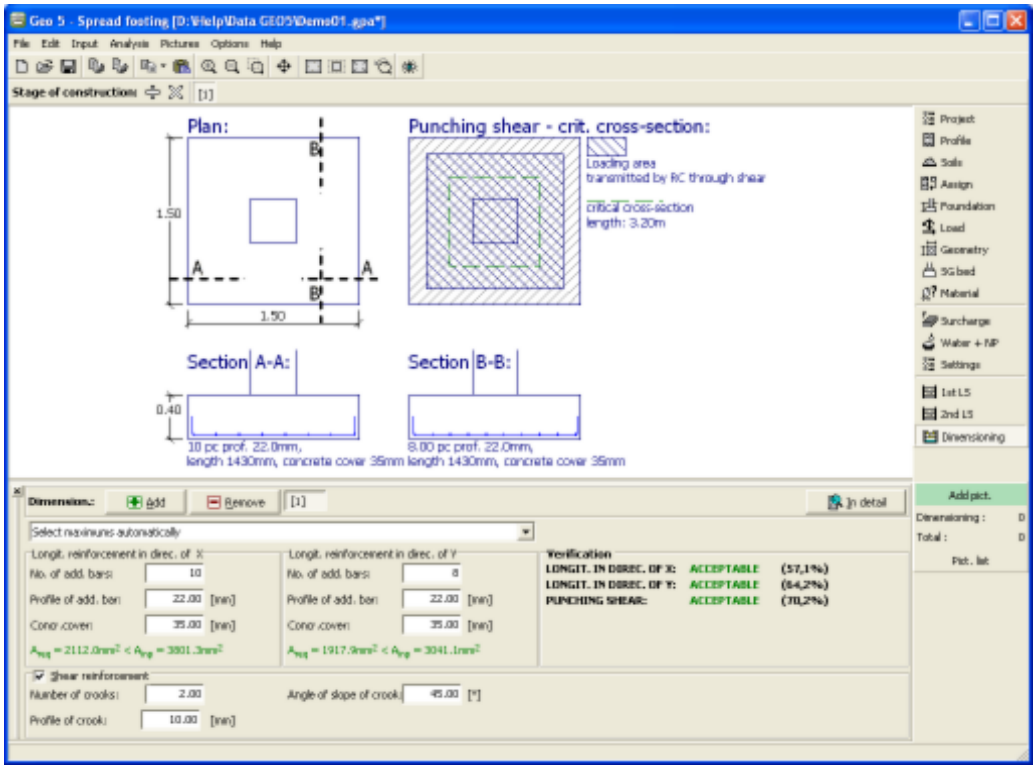
Dimensioning

The "Dimensioning" frame allows for designing and verifying the longitudinal reinforcement of a foundation and also for verifying the foundation against being pushed through. The verification can be performed either for individual loads or the program finds the most critical one (can be selected from a combo list).

The program derives the stress in the construction joint (the stress diagram is inputted in the frame "Project") and determines the internal forces in individual cross-sections. Dimensioning of the steel-reinfoced concrete structure is performed according to the standard set in the frame "Project".

The resulting information are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "In detail" button opens the dialogue window that contains detailed listing of the dimensioning results.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Dimensioning"

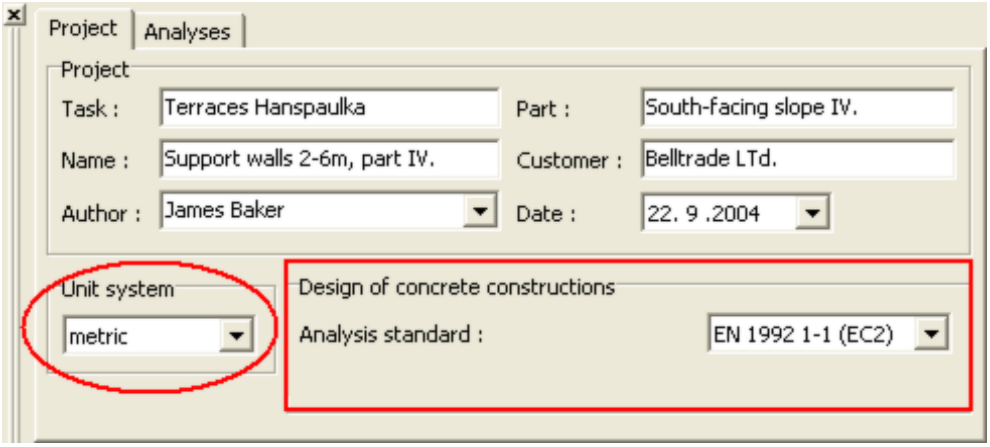
Program Pile

Project

The "Project" frame is used to input the basic project data and to specify the overall setting of the analysis run.

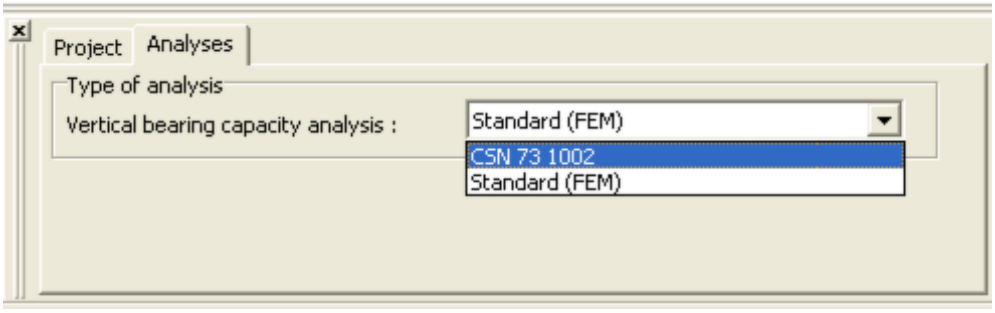
The "Project" tab sheet contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc.

The "Project" tab sheet further serves to specify a standard for concrete structures dimensioning. Referring to the selected standard the types of concrete and steel are then inputted in the "Material" and the dimensioning is verified in the "Horizontal bearing capacity" frame.



Frame "Project" - tab sheet "Project"

The "Analysis" tab sheet serves to choose the analysis of vertical bearing capacity of a pile – analysis based on CSN 73 1002, or analysis using the finite element method.



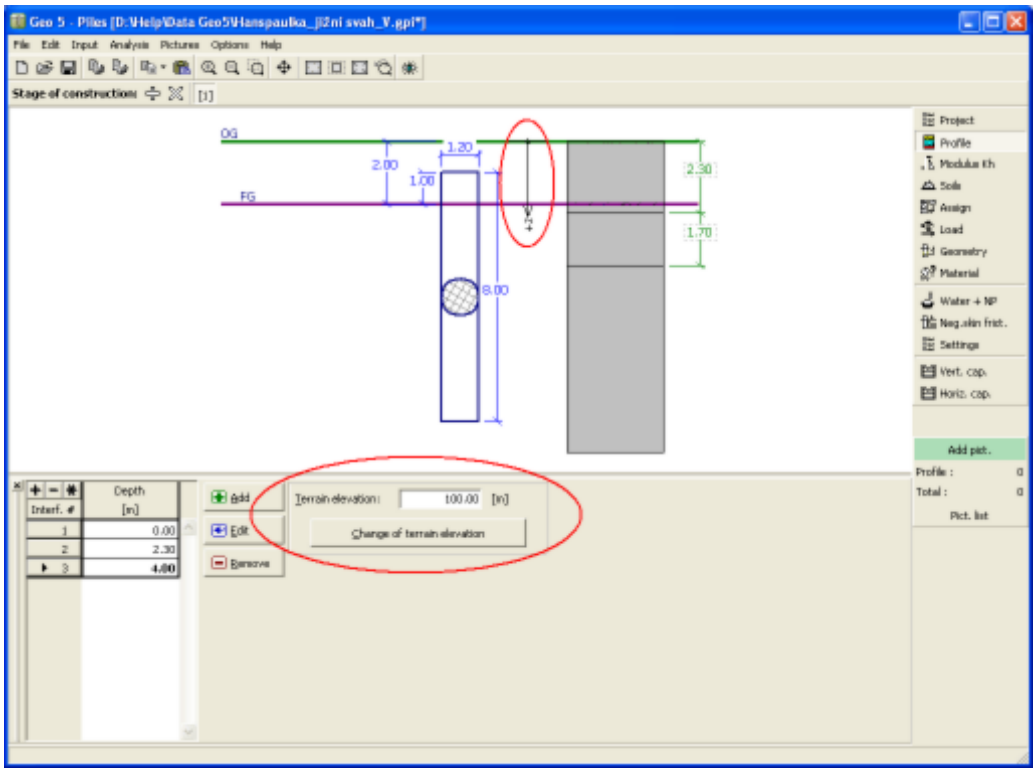
Frame "Project" - tab sheet "Analysis"

Profile

The "Profile" frame contains a table with a list of inputted interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "Add (edit) interface" dialogue window. The z-coordinate measured from the top point of a structure is specified.

The program allows for raising or lowering the top point of a structure in the "Change terrain elevation" dialogue window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".

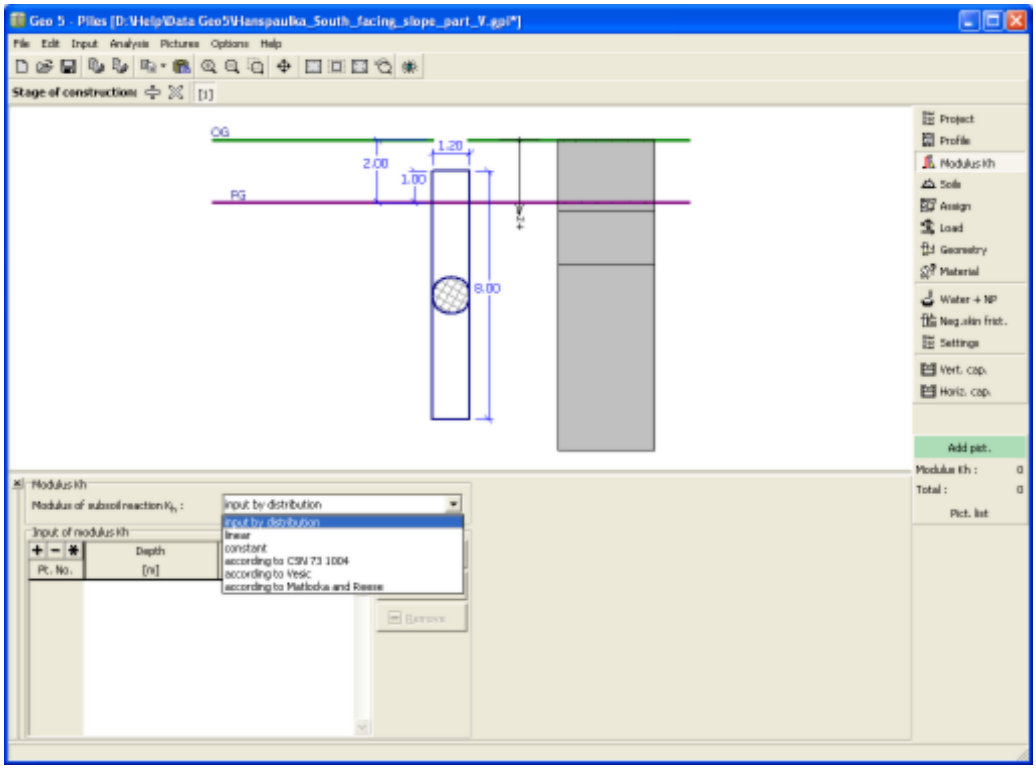


The frame "Profile"

Modulus of subsoil reaction

The combo list serves to select one of the methods for the evaluation of modulus of subsoil reaction – the required material parameters of soils are inutted in the frame "Soils" based on the selected method.

Selecting the option "Input by distribution" opens a table that allows for specifying the values of the modulus of subsoil reaction along the pile.



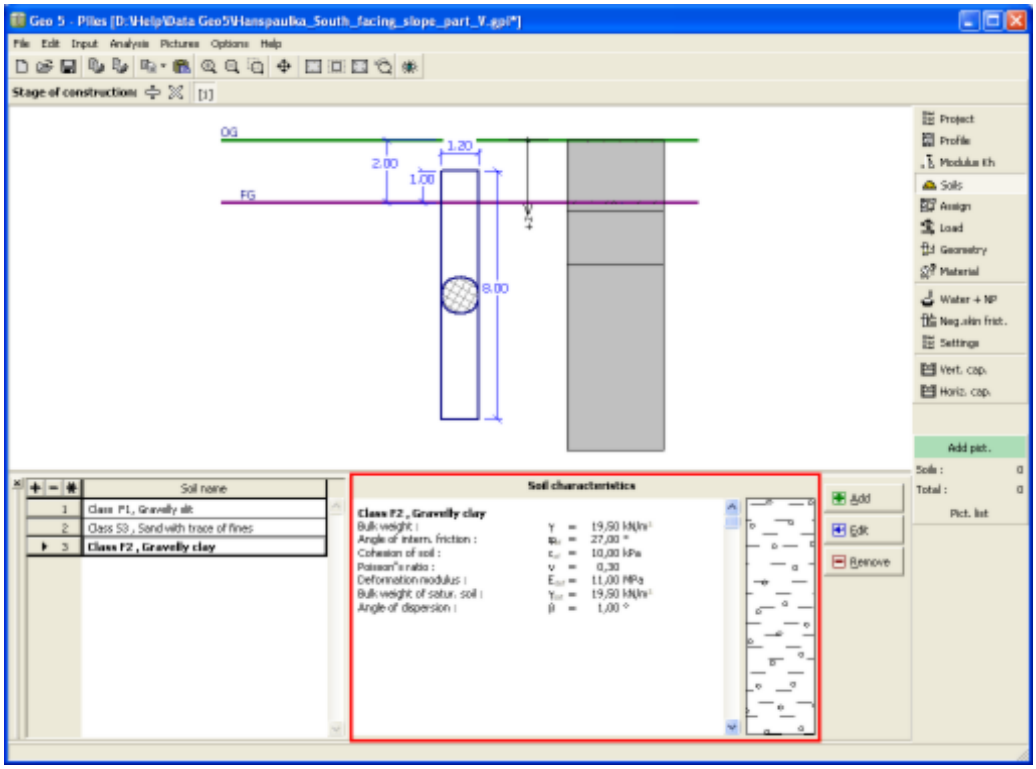
Frame "Modulus of subsoil reaction"

Soils

The "Soils" frame contains a table with a list of inputted soils. The table also provides information about currently selected soil displayed in the right part of the frame.

Adding (editing) a soil is performed in the "Add (edit) soil" dialogue window.

The soil characteristics are specified in the program "Earth Pressure". These characteristics are further specified in chapters: "Uplift pressure", "Oedometric modulus" and "Modulus of subsoil reaction". The specified soil parameters depend on the set up of modulus of subsoil rection and selected theory of analysis specified in the frame "Project", tab sheet "Analysis".

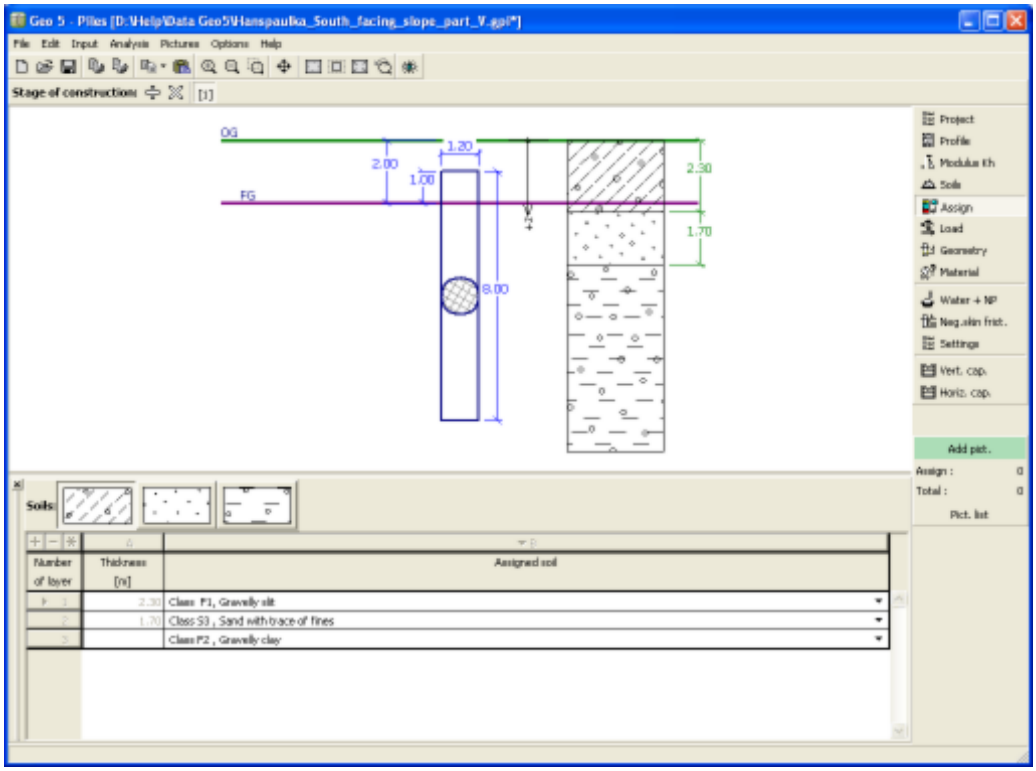


Rám "Zeminy"

Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

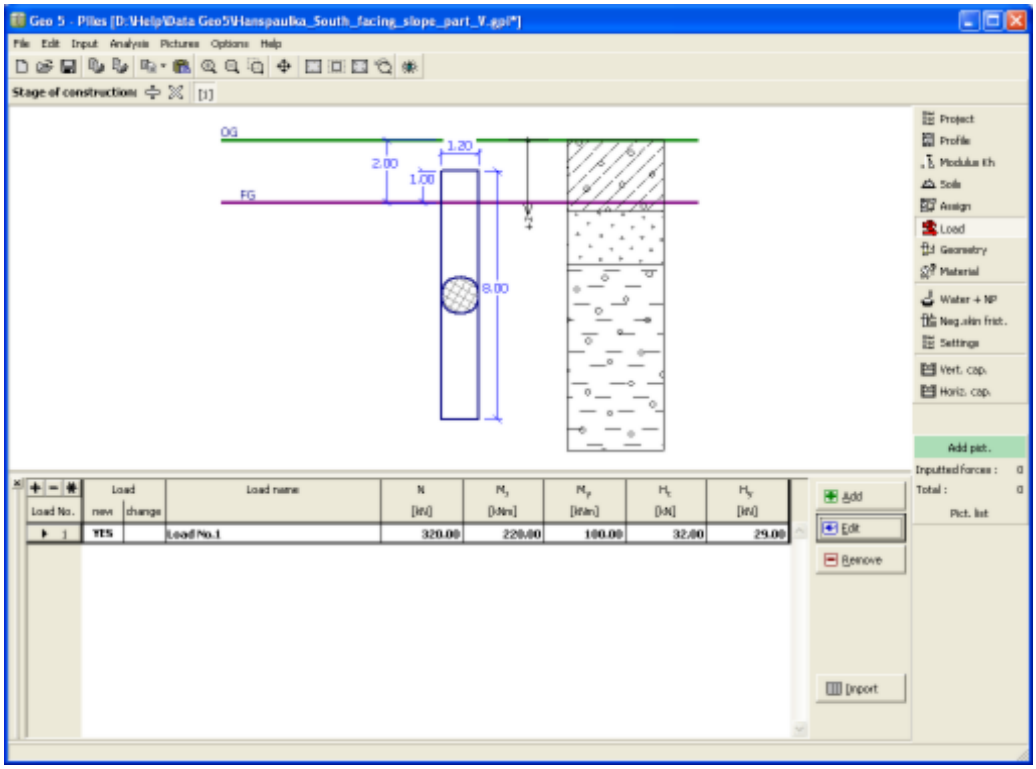
Procedure to assign soil into a layer is described in details herein.



Frame "Assign"

Load

The "Load" frame contains a table with a list inputted loads. Adding (editing) load is performed in the "New (edit) load" dialogue window. The forces are inputted following the sign convention displayed in the upper part of the dialogue window.



Frame "Load"

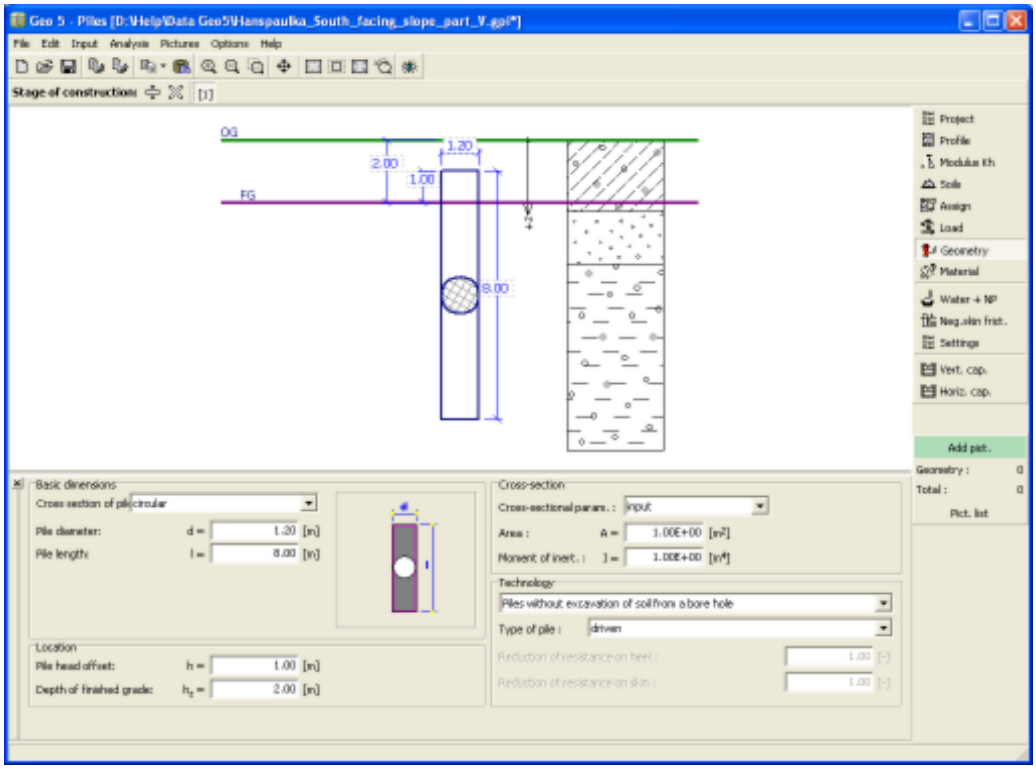
Geometry

The "Geometry" frame allows for specifying the pile cross-section (circular, circular variable, rectangle, I-type cross-section) based on the theory of analysis (specified in the "Project" frame, tab sheet "Analyses"). The selected shape with graphic hint is displayed in the central section of the frame. Input fields serve to specify dimensions of the selected cross-section.

Cross-sectional characteristics (area and moment of inertia) are computed by default, but they can also be specified (tubes, hollow cross-sections, steel I-profiles).

The bottom part of the frame serves to specify the pile location (pile lift out and the depth of finished grade). The pile lift out can also be negative – in such a case the pile is placed "in-ground".

Providing the pile is analyzed using the finite element method it is possible to account for influence of pile technology by selecting the specific type of pile or directly by inputting coefficients.



Frame "Geometry"

Material

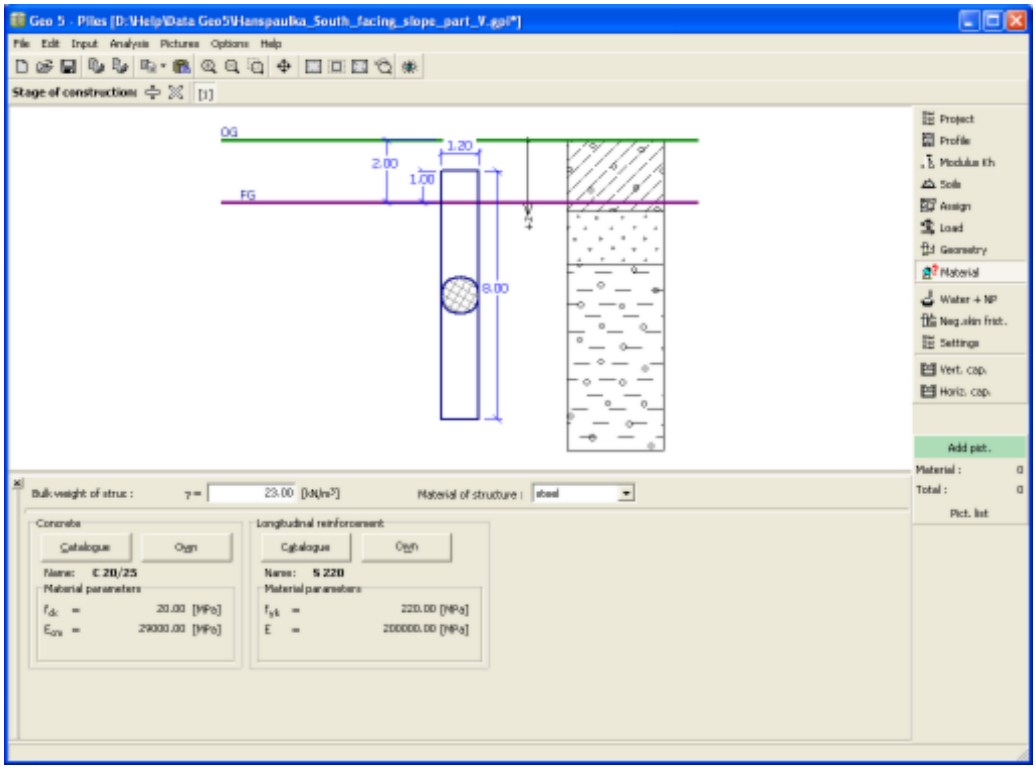
The "Material" frame allows for specifying the material parameters. The bulk weight of a structure and material of a pile (concrete, timber, steel) are introduced in the input field in the right part of the frame.

The elastic and shear moduli need to be specified when assuming timber or steel piles.

In case of a concrete pile the concrete material and parameters of transverse and longitudinal steel reinforcements are required. Two options are available when selecting the type of material:

- the "Catalogue" button opens the "Material catalogue" dialogue window (for concrete or steel reinforcements), the list of materials then serves to select the desired material
- the "User" button opens the "Edit material – concrete" dialogue window (for concrete) or the "Edit material – concrete steel" dialogue window (for longitudinal and transverse steel reinforcements), which allows for manual specification of material parameters

The catalogues content depends on the selection of standard for the design of concrete structures set in the "Project" frame.



Frame "Material"

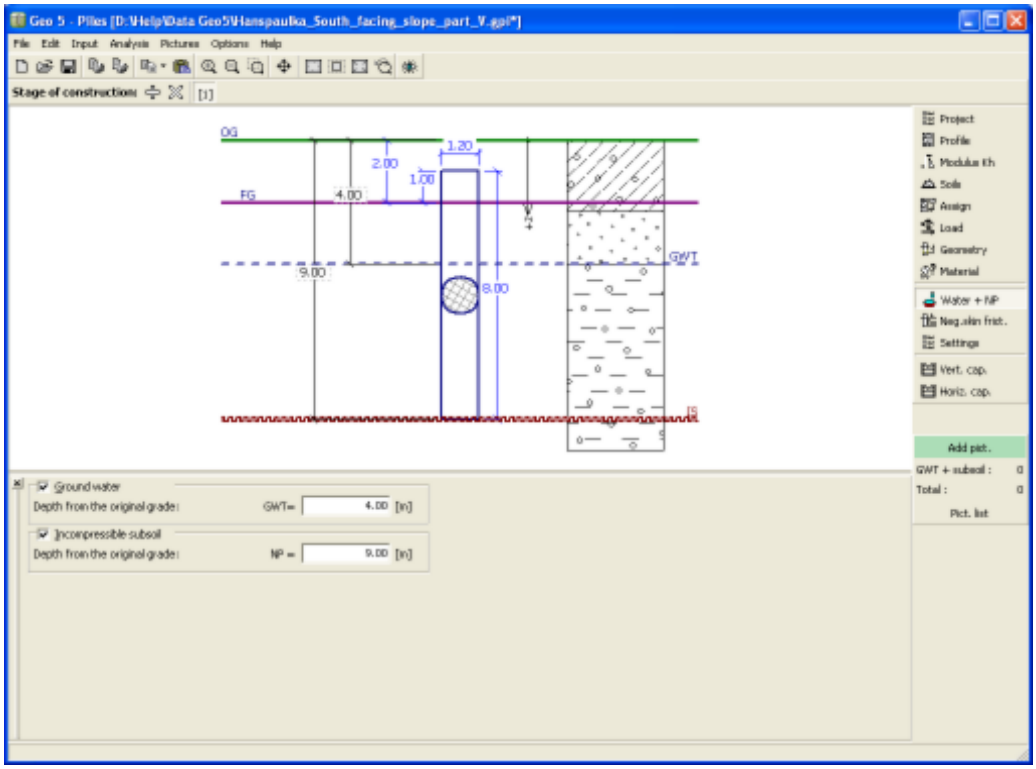
Water, incompressible subsoil

The "Water + IS" frame serves to specify the depth of ground water table and level of incompressible subsoil.

The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The influence of water is manifested by the change of ground water pressure.

The incompressible subsoil cuts off the influence zone below foundation and also influences reduction in settlement.

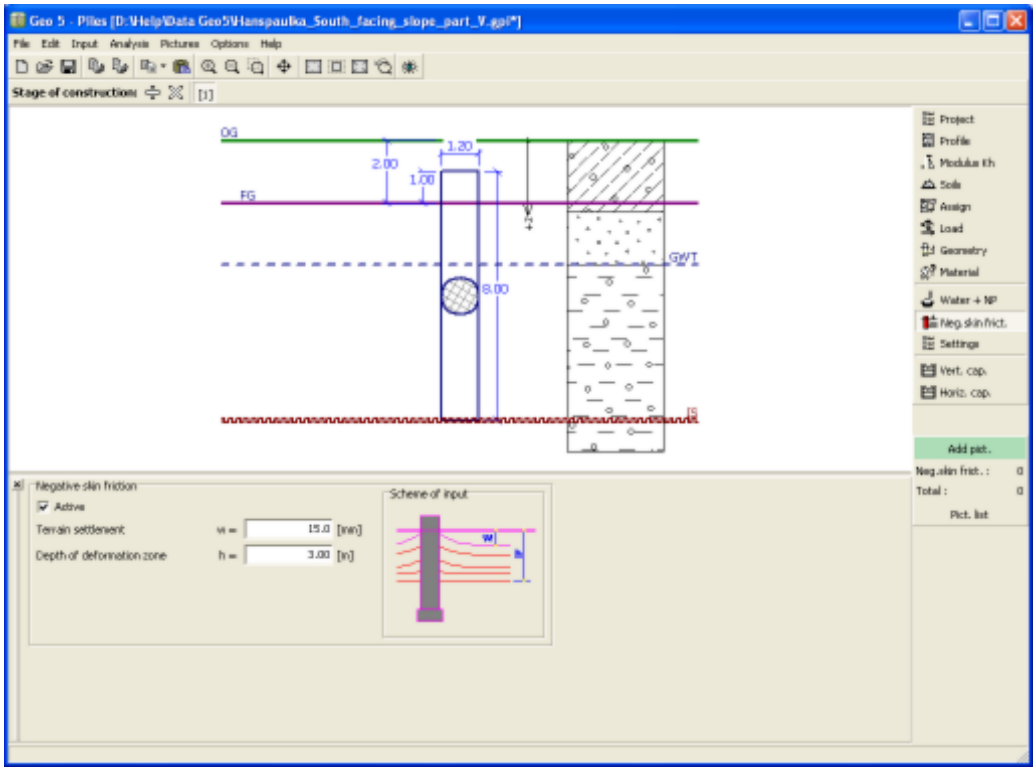


Frame "Water, incompressible subsoil"

Negative skin friction

The "Negative skin friction" frame serves to specify the settlement of surrounding terrain and the depth of influence zone. For more information on the influence of negative skin friction the user is referred to theoretical section.

The setting option in the frame is active only when the finite element method is selected for the analysis in the frame "Project" – tab sheet "Analysis".



Frame "Negative skin friction"

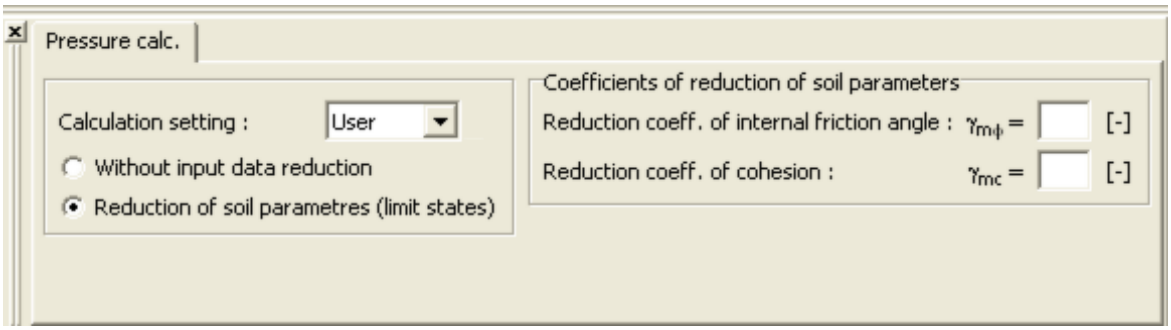
Setting

The "Setting" frame contains basic settings for footings verification. The program offers pre-setting for different countries (Czech Republic, Germany, France,...) and "Standard setting" recommended by the authors of the program for countries no included in the list. While changing settings in the combo list the values of coefficients of reduction of soil parameters in corresponding windows are changed.

An arbitrary analysis setting is available with the option "User setting".

Selecting the option "Reduce soil parameters" allows for specifying individual values of the coefficients of reduction of soil parameters (e.g., recommended values according to EC7).

The setting in the frame becomes available only if the type of analysis FEM is selected in the frame "Project" – tab sheet "Analysis".


Pressure calc.
Calculation setting :
User
Without input data reduction
Reduction of soil parametres (limit states)
Coefficients of reduction of soil parameters
Reduction coeff. of internal friction angle : $\gamma_{m\varphi}$ =
[-]
Reduction coeff. of cohesion : γ_{mc} =
[-]

Frame "Setting"

Vertical bearing capacity

The "Vertical bearing capacity" frame has two variants depending on the type of analysis of pile vertical bearing capacity:

- according to ČSN 73 1002
- using the finite element method

Vertical bearing capacity CSN

The "Vertical bearing capacity" frame serves to the vertical bearing capacity of a pile. Several analyses can be carried out in the frame. The verification can be performed for individual loads, or the program locates the most critical one (can be selected from a combo list).

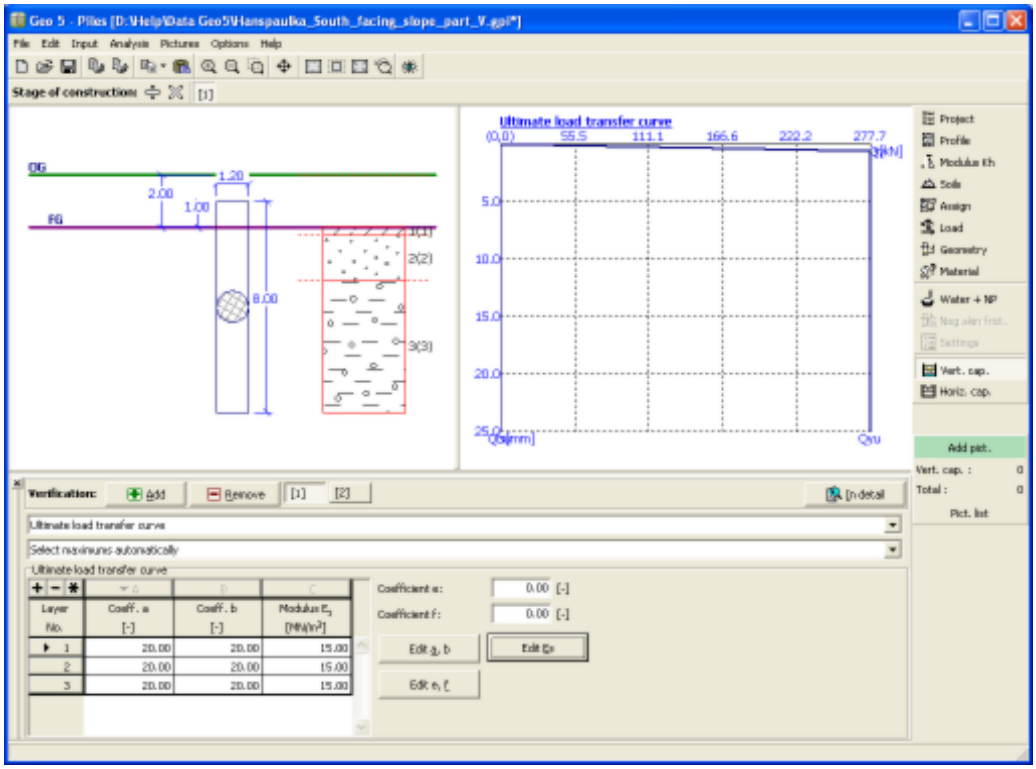
The analysis is carried out according to one of the method based on ČSN 73 1002. Parameters needed in individual methods are inputted in the lower part of the frame.

The "Loading limit curve" and "Point-bearing pile capacity" modes contain a table in the lower part of the frame, which allows for direct editing of inputted parameters using mouse. The "Edit" buttons open the dialogue windows that include hints for individual parameters. A given dialogue window allows for storing inputted parameters for a given layer in the table by pressing the "OK" button.

Analysis results can be displayed in the right part of the desktop.

The "In detail" button opens the dialogue window containing detailed listing of the verification results.

The analysis results are displayed in the right part of the desktop. Visualization of the results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Vertical bearing capacity - analysis according CSN 73 1004"

Vertical bearing capacity FEM

The "Vertical bearing capacity" frame allows for verifying the pile vertical bearing capacity. The analysis is performed automatically when swtiching to this frame. More computations can be performed in the frame. The verification can be performed either for individual loads or the program finds the most critical one (can be selected from a combo list)

The analysis is performed with the help of finite element method. The results are automatically updated whenever one of the analysis parameters "Maximal deformation", " Coefficient increasing limit skin friction due to technology" or "Procedure determining influence zone bellow heel" is changed.

Two options are available to determine influence zone below the heel:

By default the analysis follows the procedure described in the theoretical part of the hint in secton "Depth of influence zone".

The second options assumes the depth influence zone to be set a k-th multiple of the pile diameter. During gradual increase of pile surcharge the depth of influence zone is continuously changed from zero at the onset of loading to the k-th multiple of the pile diameter when exceeding the total load.

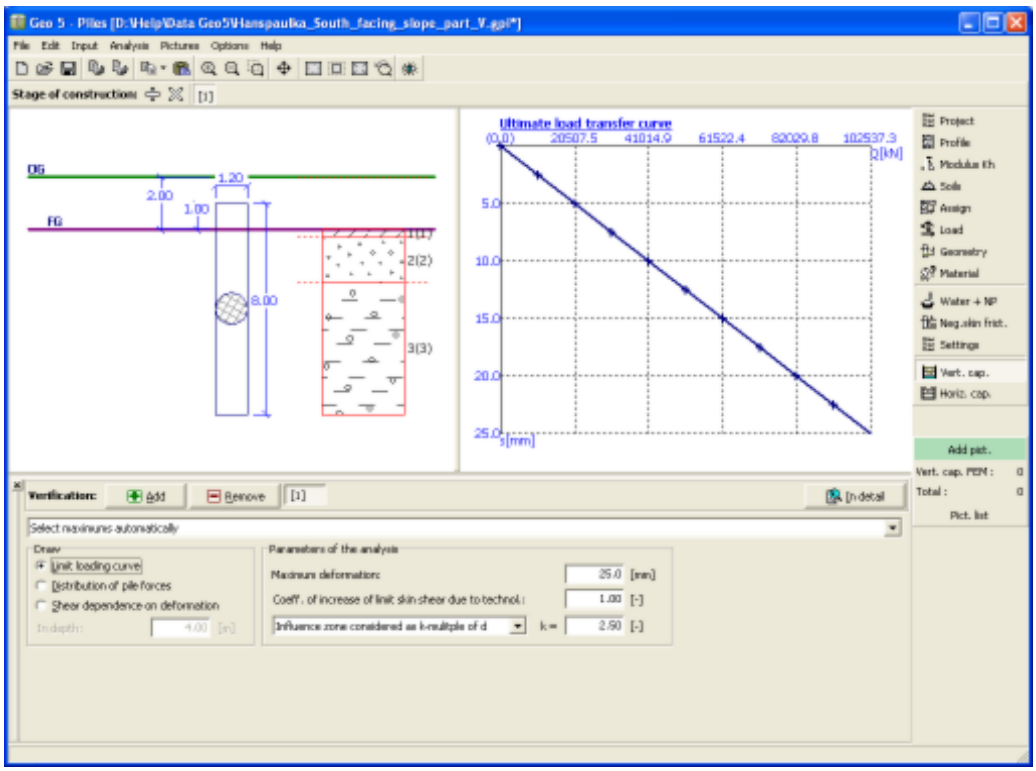
The second method, originaly used in the old verision GEO4, with the value of k= 2,5 offers less accurate results and usually underestimates the pile bearing capacity. Therefore a new option that allows for specifying the depth of influnece zone through anlsysis is offered and is also recommeded by default setting.

Switching between results is avaiable in the left part of the frame (limit loading curve, distributions of internal forces, dependence of shear on displacement). The shear-displacement relationship is derived for a given depth measured from the pile head. The results are updated whenever the depth is changed.

The "In detail" button opens the dialogue window, which contains detailed listing of the results of verification analysis.

VýsledkyVisualization of results can be adjusted in the "Setting visualization style" dialogue

window.



Frame "Vertical bearing capacity - analysis according FEM"

Horizontal bearing capacity

The horizontal bearing capacity of a pile is verified in the "Horizontal bearing capacity" frame. Several analyses can be carried out. The verification analysis can be carried out for individual loads, prescribed displacements, or the program finds the most critical load (can be selected from a combo list). Assuming the prescribed displacement type of load requires introduction of boundary conditions in pile head (translation and rotation).

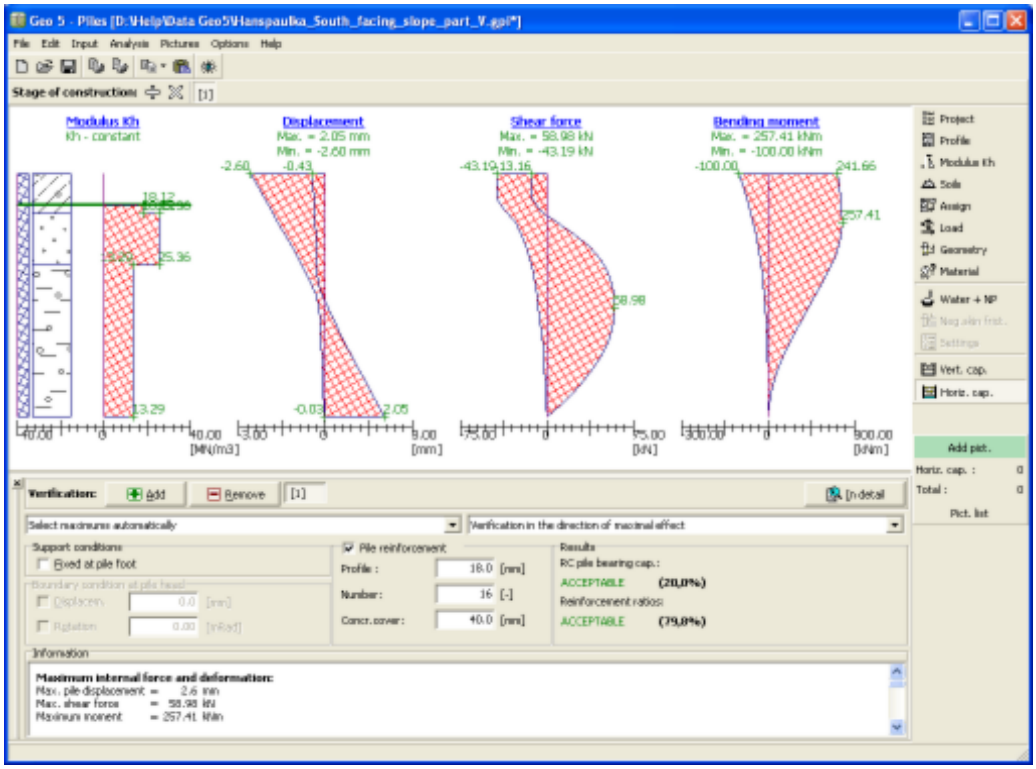
The fixed end type of boundary condition prescribed in the pile heel can be assumed for all types of loading.

For steel reinforced concrete pile the programs allow for verifying the reinforcement based on the standard selected in the frame "Project".

The combo list serves to specify the direction of pile verification (x,y); for a circular pile the program allows for displaying the results in the most stressed direction.

The "In detail" button opens the dialogue window that contains detailed listing of the verification results.

The analysis results are displayed on the desktop. Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Horizontal bearing capacity"

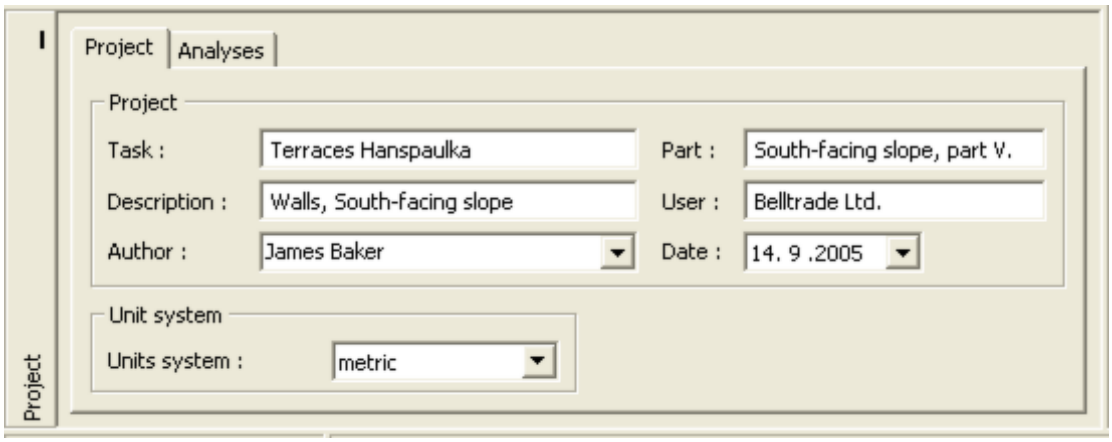
Program Settlement

Project

The "Project" frame is used to input the basic project data and to specify the overall setting of the analysis run. The "Project" tab sheet contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc.

The "Project" tab sheet also allows the user to switch analysis units (metric / imperial).

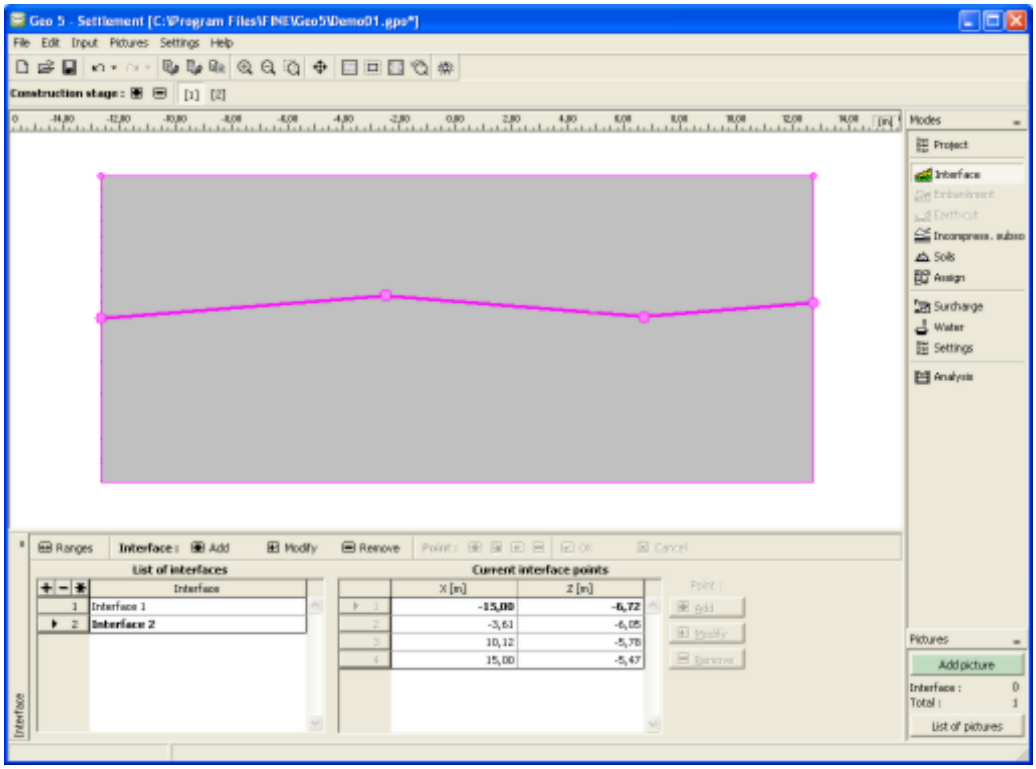
The frame "Analyses" serves to select the theoretical approach for the settlement analysis and the way of reducing the influence zone.



Frame "Project" - tab sheet "Project"

Interface

The "Interface" frame serves to introduce individual soil interfaces into the soil body. Detailed description how to deal with interfaces id described herein.

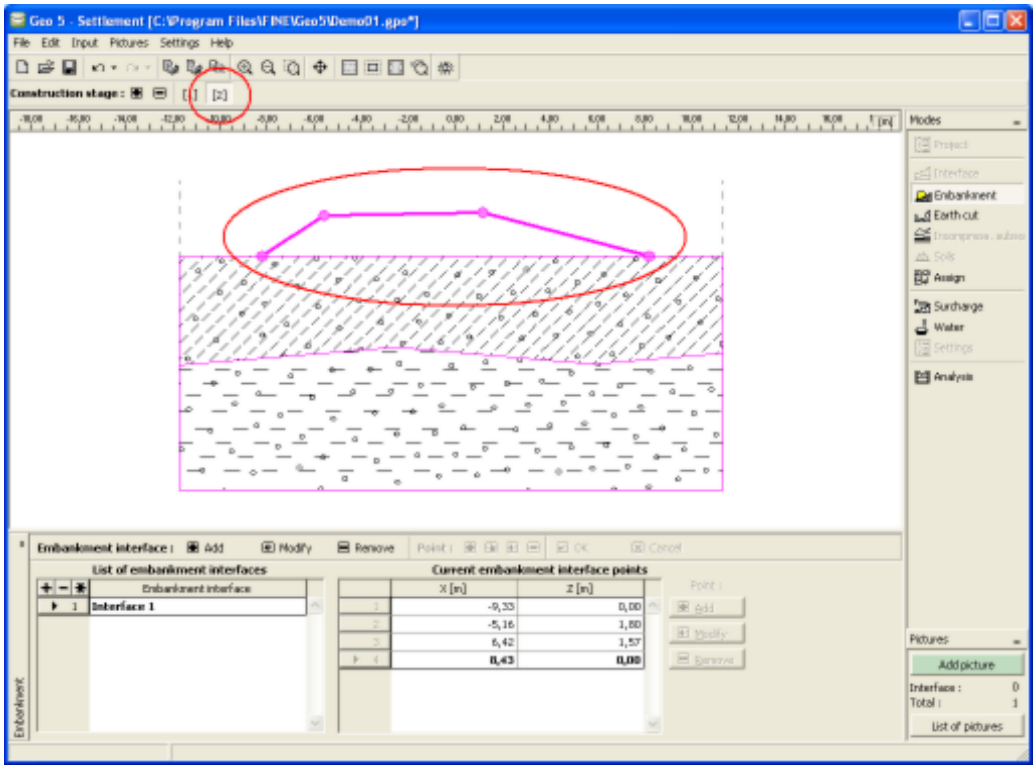


Frame "Interface"

Embankment

The "Embankment" frame allows for inputting interfaces to create an embankment above the current terrain. The frame contains a table with a list of interfaces forming the embankment. A table listing the points of currently selected interface of the embankment is displayed in the mid section of the frame. Inputting an embankment interface follows the same steps as used for standard interfaces.

An embankment cannot be specified in the first stage of construction. An embankment cannot be built if there is an earth cut already specified in a given stage - in such a case either a new stage of construction must be introduced for embankment input or the existing open cut must be first removed.



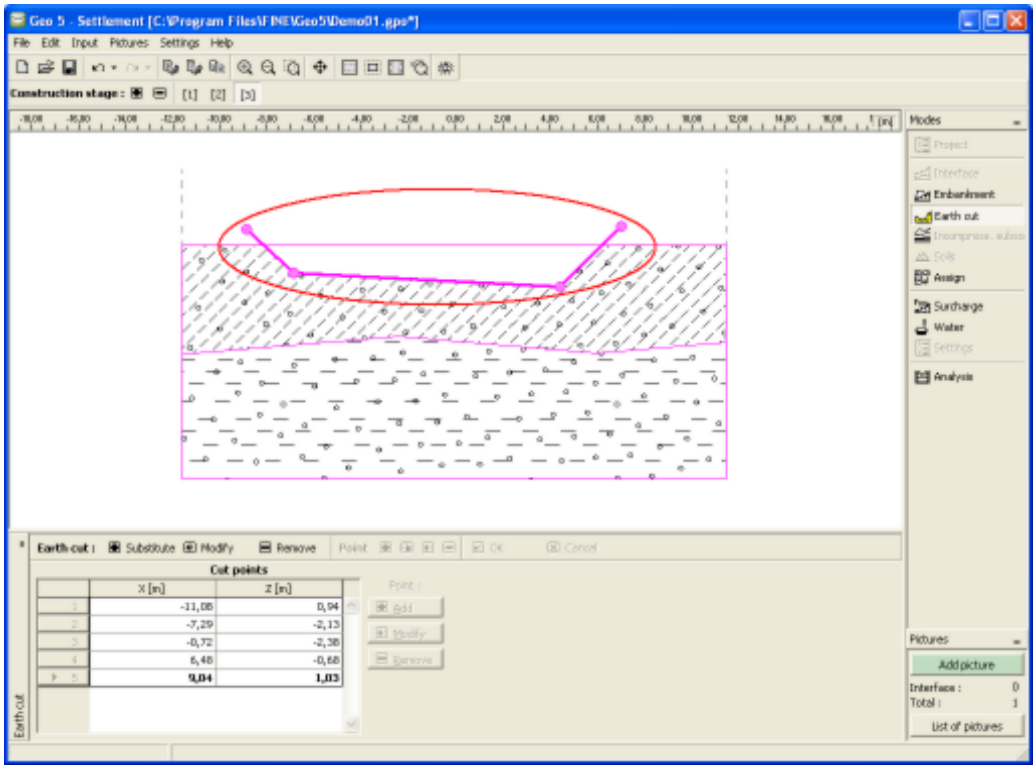
Frame "Embankment"

Earth cut

The "Earth cut" frame serves to specify the shape of an open cut. This function allows for modifying the terrain profile within a given stage of construction. Several earth cuts can be introduced at the same time. In such a case some of the lines in the cut appear partially above the terrain.

A table listing individual interface points is displayed in the left part of the frame. Inputting an earth cut interface follows the same steps as used for standard interfaces.

An open cut cannot be specified in the first stage of construction. An earth cut cannot be built if there is an embankment already specified in a given stage - in such a case either a new stage of construction must be introduced for earth cut input or the existing embankment must be first removed.



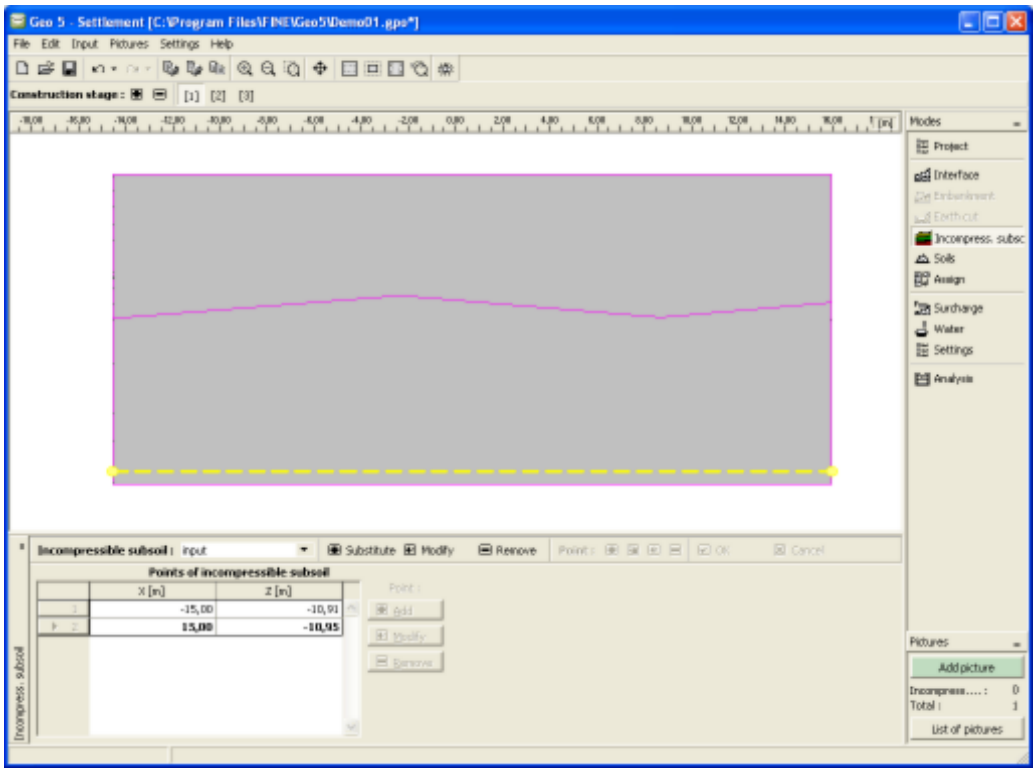
Frame "Earth cut"

Incompressible subsoil

The frame "Incompressible subsoil" serves to input a depth of incompressible subsoil.

Inputting the depth of incompressible subsoil is the same as when inputting standard interfaces.

Inputting an incompressible subsoil is one the options how to restrict an influence zone – if inputted, then both ranges and tilted sections are drawn up to a depth of incompressible subsoil. No ground deformation appears below the incompressible subsoil.



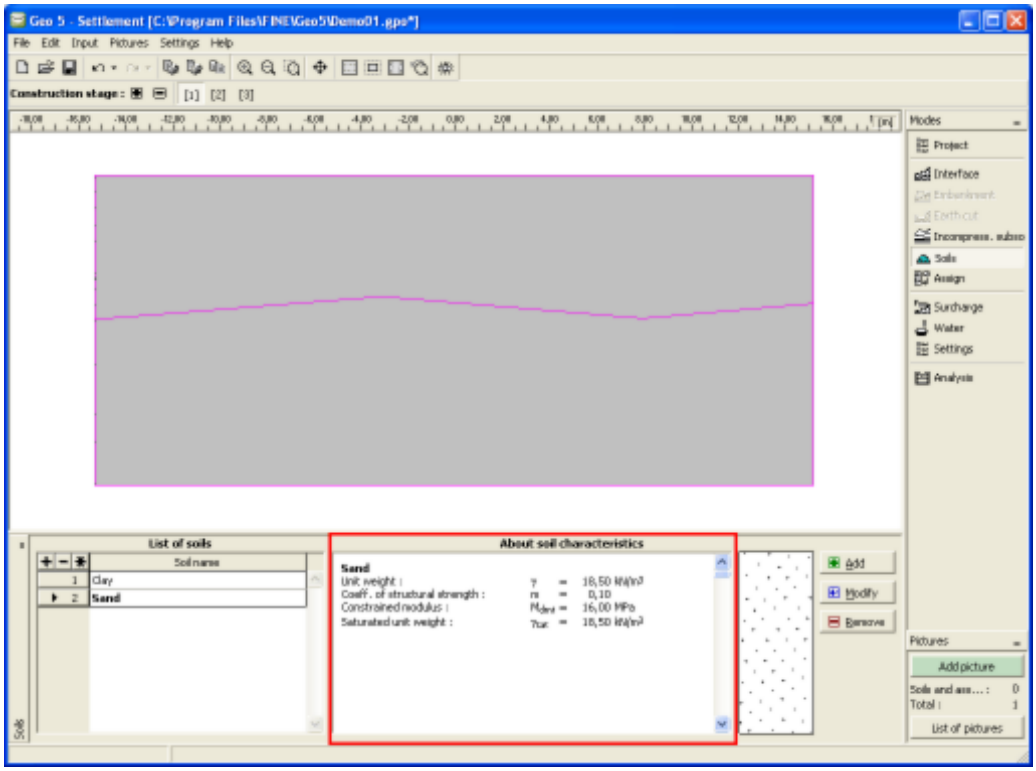
Frame "Incompressible subsoil"

Soils

The "Soils" frame contains a table with a list of inputted soils. The table also provides information about currently selected soil displayed in the right part of the frame.

Adding (editing) a soil is performed in the "Add (edit) soil" dialogue window.

The soil characteristics are specified in the program "Settlement". These characteristics are further specified in chapters: "Uplift pressure" and "Settlement analysis" - the input parameters of soils are determined based on the selected theory of analysis in the frame " Project" tab sheet "Analyses".

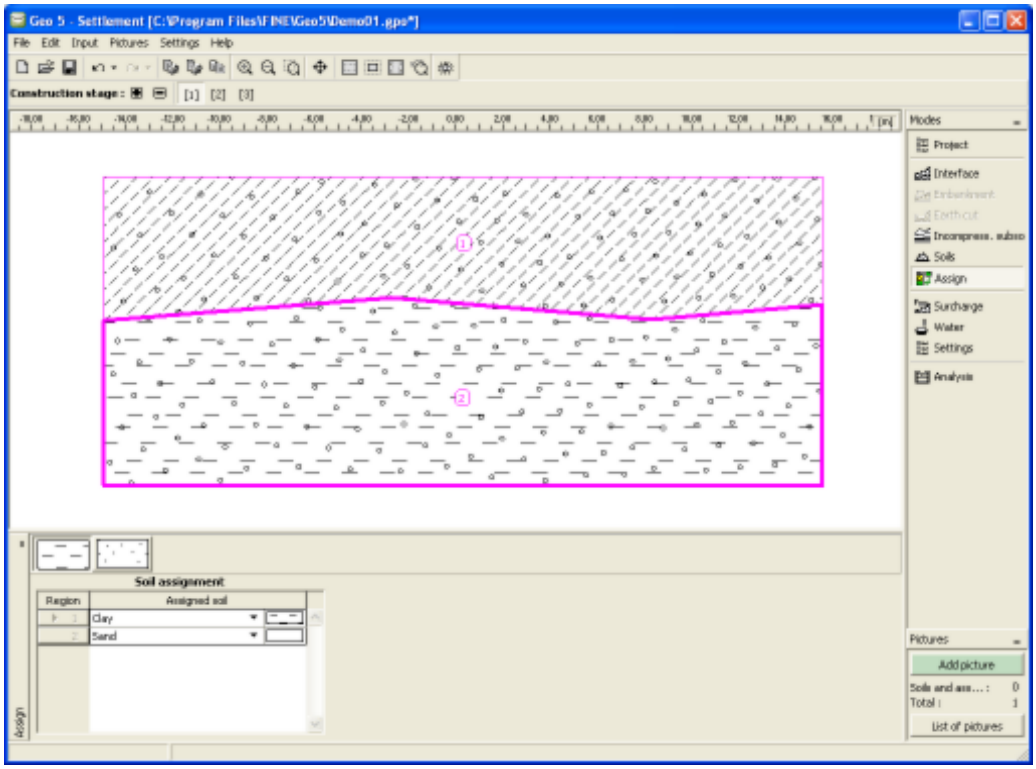


Frame *"Soils"*

Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

Procedure to assign soil into a layer is described in details herein.



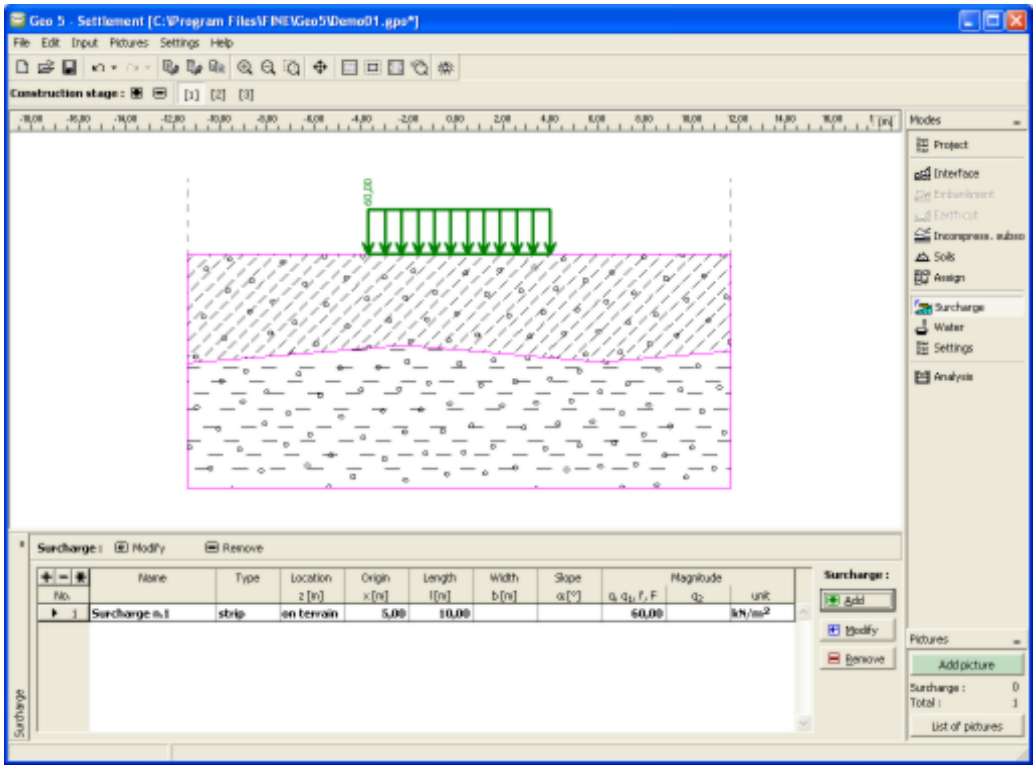
Frame "Assign"

Surcharge

The "Surcharge" frame contains a table with a list of inputted surcharges. Adding (editing) surcharge is performed in the "New (edit) surcharge" dialogue window. The inputted surchages can be edited on the desktop with the help of active dimensions or active objects, respectively.

All inputted parameters of a surcharge can be modified in the construction stage where the surcharge was specified. Only the surcharge magnitude can be modified in all subsequent construction stages (option "Adjust surcharge").

Influence of surcharge on the change of stress in the soil body is described in the theoretical part of the help section.



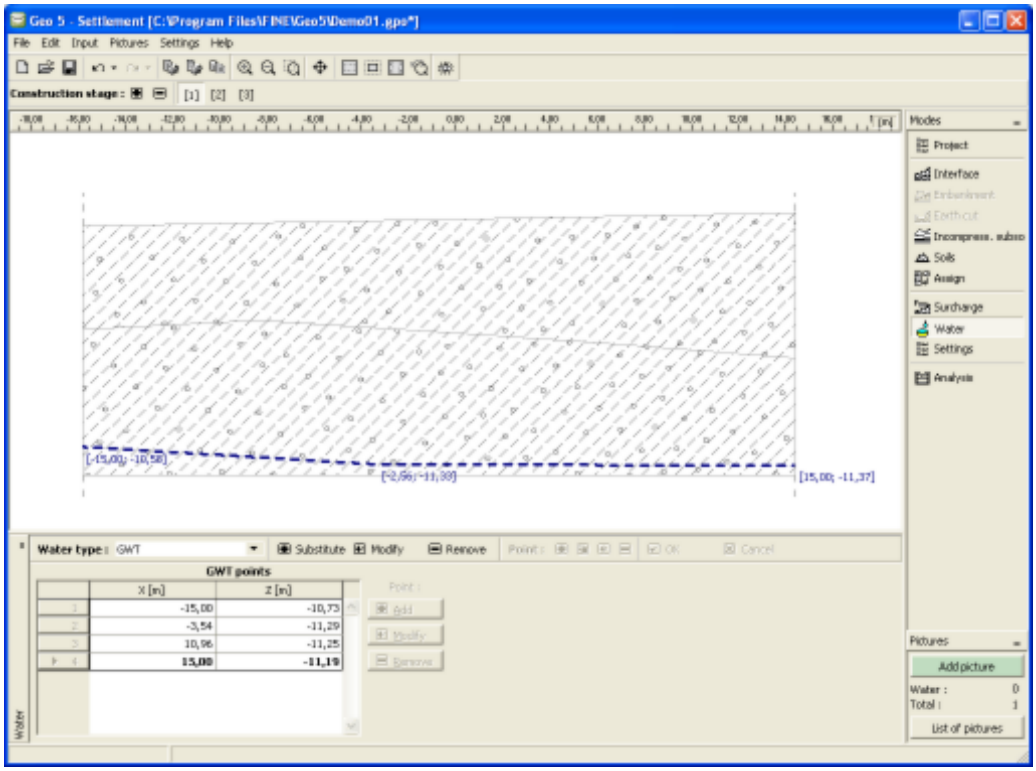
Frame "Surcharge"

Water

The "Water" frame serves to set the type of ground water table.

Inputting the ground water table or isolines, respectively, is identical with the standard input of interfaces.

If the inputted data in individual stages are different, the program then allows for accepting the data from the previous stage of construction by pressing the "Accept" button.



Frame "Water"

Setting

The frame "Setting" allows for specifying the position of control holes and thicknesses and locations of layers where the stress values are calculated.

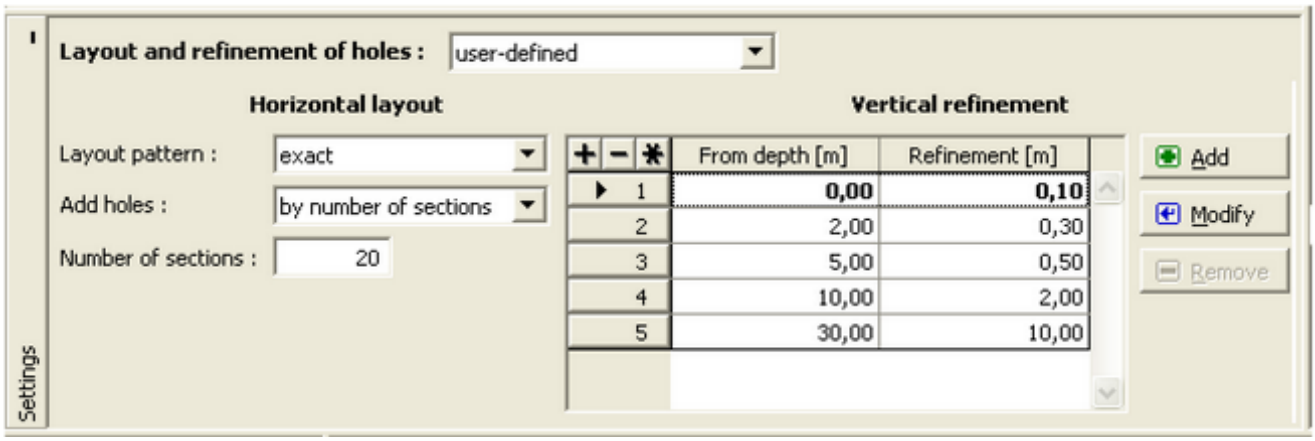
The program determines stresses at individual control holes. The terrain is always subdivided into twenty holes with even spacing. Additional holes are automatically generated in points specifying terrain, embankment, GWT, soil layer interfaces and end points of surcharge. The control (calculation) holes can be plotted in the frame "Analysis".

Individual holes are divided into layers according to the inputted values. The first layer always coincides with the original ground. In addition, all points specifying interfaces, GWT and incompressible subsoil are included. The default setting of thicknesses of layers ensures reasonable speed and accuracy of the analysis.

The layers are introduced up to depth of 250m. In actual analyses, however, the depth of influence zone is restricted either by the inputted incompressible subsoil or by the reduction of magnitude of stress change or by the structural strength, respectively (depending on the setting in the frame "Project").

The number and location of calculation holes can be adjusted when selecting the option "user defined". In such a case it is possible to select both the location of doles and thicknesses and location of layers. The holes are then created according to the input – in addition, the program automatically includes all important points. When selecting the option exact distribution, the holes are included into all terrain points, soil layer interfaces, embankments, GWP and into end points of surcharge. When selecting the option minimal distribution, the holes are not included into points of interfaces of soil and embankment layers.

For standard analyses we recommend to keep the default setting of the analysis.



Frame "Setting"

Analysis

The "Analysis" frame displays the analysis results. The analysis is carried out based on the calculation theory selected in the frame "Project", tab sheet "Analyses". The depth of influence zone is determined either by inputted incompressible subsoil, by the theory restriction of primary stress or by the theory of structural strength.

Information regarding the course of analysis, maximum settlement and the depth of influence zone are printed out in the bottom section of the frame. The results, as the main output, are displayed on the screen. To view the results, use the horizontal bar in the upper section of the screen, which allows for adjusting the way the resulting values are plotted. The bar contains the following control items:

- the button to display the "Settlement - results visualization setting" dialogue window. This dialogue window allows for specifying all drawing parameters: parameters to display depression line and influence zone, to set color range, to draw tilted sections, isosurfaces and

isolines, etc.

- option to store individual views

- selecting values for visualization – either total values, or their change during the last calculation stage or their change in comparison with previous stages can be plotted. The setting is available only in problems where it makes sense. It is therefore possible to display the change of stress, settlement or deformation in comparison with previous stages – however, always the current depth of influence zone is plotted.

- selecting variables

SigmaZ,tot - overall vertical total stress [*kPa,ksf*]

SigmaZ,eff - overall vertical effective stress [*kPa, ksf*]

Pore pressure - stress due to water [*kPa, ksf*]

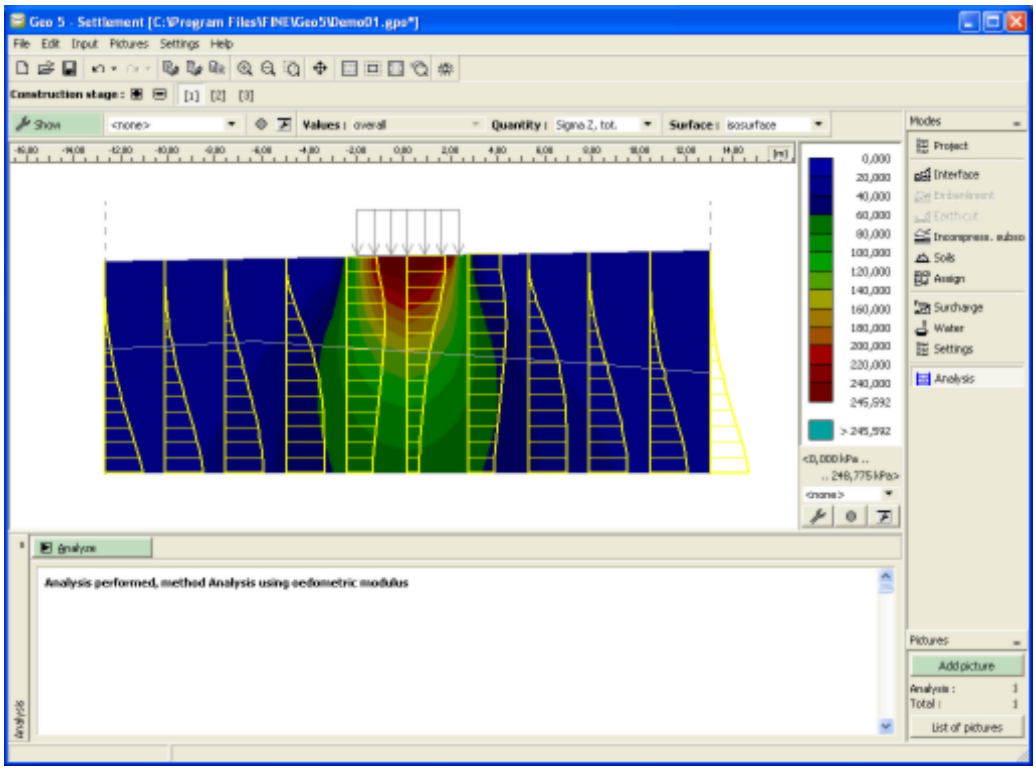
Settlement - settlement of a point [*mm, feet*]

Deformation - relative settlement of a layer [-] **1000*

- plotting option (do not plot, isosurfaces, isolines)

The color range is visible on the right part of the desktop. The buttons for setting the color range are located below.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Analysis"

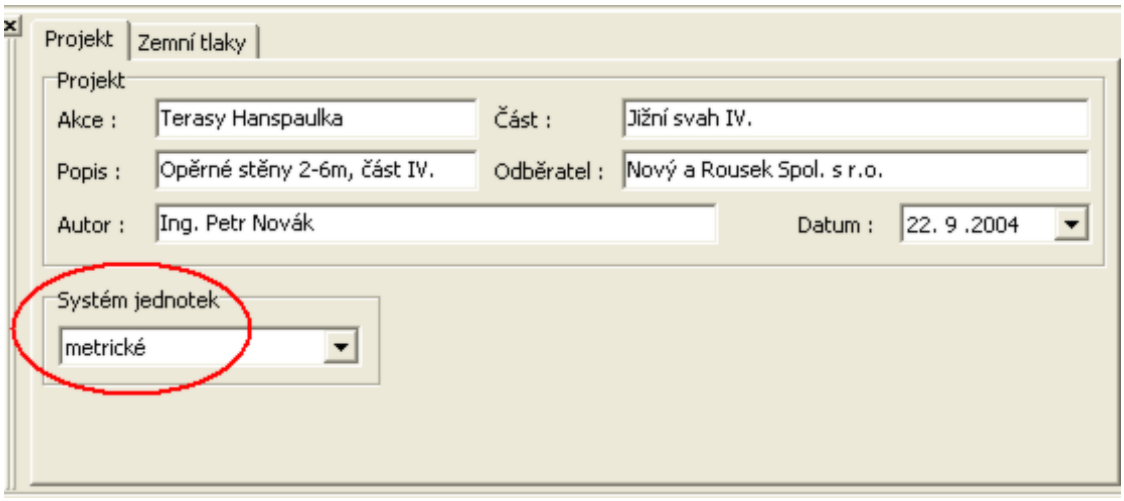
Program Svahovky

Projekt

Rám "Projekt" je určen k zadání základních dat o úloze a k celkovému nastavení výpočtů programu. Záložka "Projekt" obsahuje vstupní formulář pro zadání údajů o zadané úloze, tzn. informace o projektu, jeho popis, datum atd.

V záložce "Projekt" lze také přepínat jednotky (metrické / imperiální).

V záložce "Zemní tlaky" se volí základní teorie popřípadě normy, podle kterých bude úloha počítána.



Rám Projekt - záložka "Projekt"

Geometrie

Rám "Geometrie" obsahuje tabulku se seznamem úseků tvořených svahovkami. Pro kažký úsek se zadává množství, sklon a typ svahovky (normální, velká, vodní).

Výztuhy

Rám "Výztuhy" obsahuje tabulku se seznamem výztuh. V dialogovém okně "Nová výztuha" se pak zadávají umístění výztuhy a parametry výztuhy (délka přesahu, únosnost na vytržení a pevnost výztuhy).

Základ

V rámu "Základ" se zadávají údaje o základu pod svahovkami. Základ může být buď betonový nebo tvořený štěrkopískovým polštářem. V tomto případě se v rozbalitelném seznamu vybírá přiřazená zemina (rozbalitelný seznam obsahuje zeminy zadané v režimu "Zeminy"). Je-li základ tvořen zeminou, je nutné zadat únosnost zeminy Rd - posouzení únosnosti základové půdy pod svahovkami je pak provedeno v rámu "Dimenzace" pro první svahovku.

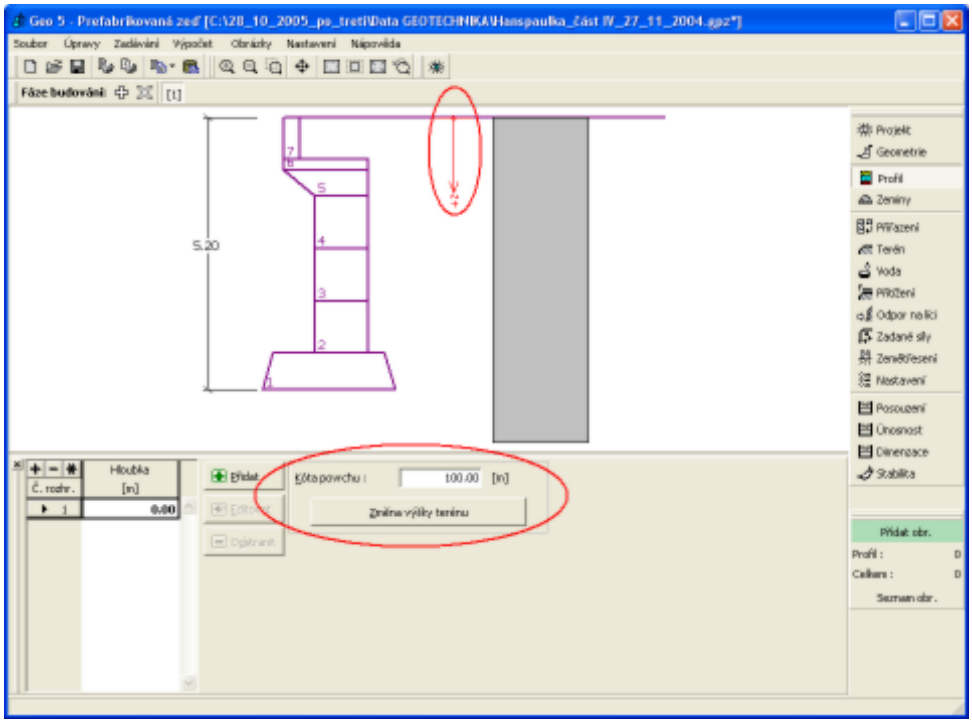
V tomto rámu lze také zadat parametry odvodnění. Toto zadání nemá žádný vliv na prováděné výpočty - je v programu k dispozici pouze pro účely dokumentace.

Profil

Rám "Profil" obsahuje tabulku se seznamem zadaných rozhraní. Po zadání rozhraní lze na desktopu editovat mocnosti jednotlivých vrstev pomocí aktivních kót.

Přidání (editace) rozhraní se provádí v dialogovém okně "Přidání (editace) rozhraní". Zadává se z-ová souřadnice od horního bodu konstrukce (viz osa z).

Program umožňuje zvýšit resp. snížit horní bod konstrukce v dialogovém okně "Změna výšky terénu" a tím posunout celé rozhraní při zachování mocností jednotlivých vrstev. Tato funkce je důležitá především při převzetí zemního profilu z programu "Terén".



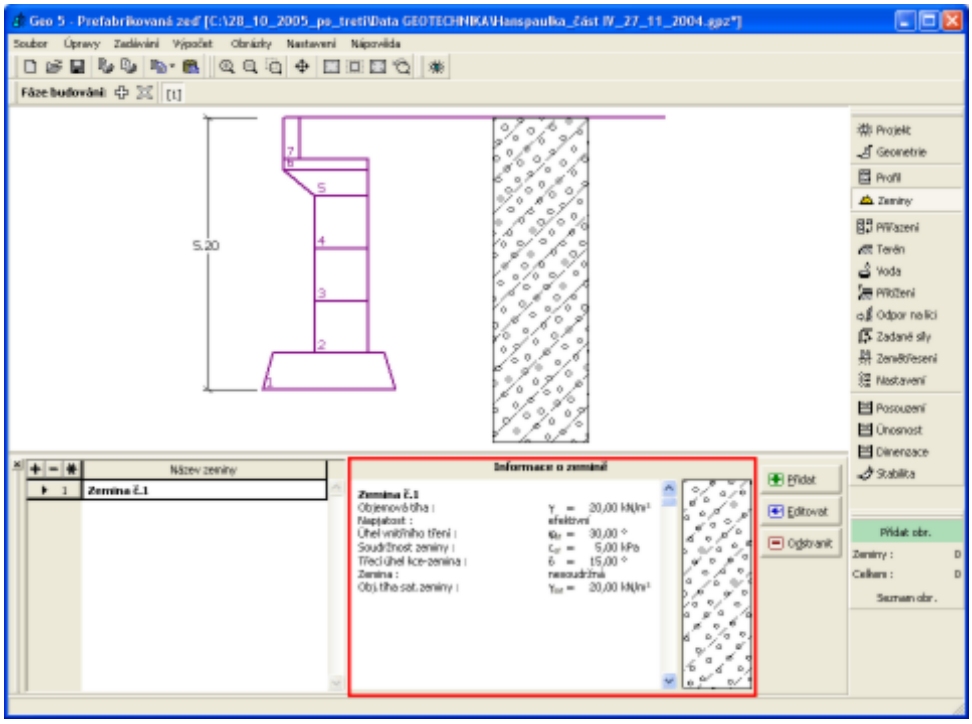
Rám "Profil"

Zeminy

Rám "Zeminy" obsahuje tabulku se seznamem zadaných zemin. Zvláštností této tabulky je zobrazení údajů o aktuálně vybrané zemině v pravé části rámu.

Přidání (editace) zeminy se provádí v dialogovém okně "Přidání (editace) zeminy".

V programu "Svahovky" se zadávají charakteristiky zemin, které jsou blíže popsané v následujících kapitolách: "Základní data", a "Vztlak".



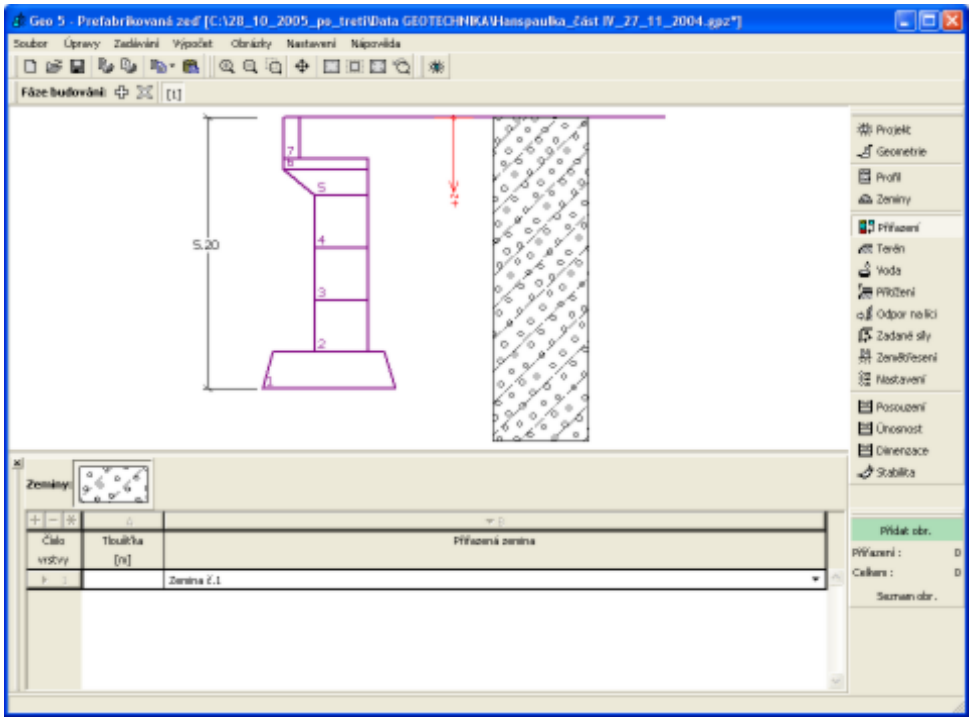
Rám "Zeminy"

Přiřazení

Rám "Přiřazení" obsahuje seznam vrstev profilu a k nim přiřazených zemin. Seznam zemin

je graficky znázorněn pomocí tlačítek v liště nad tabulkou, nebo je přístupný z rozbalitelného seznamu u každé vrstvy profilu.

Postup přiřazení zemin je podrobně popsán zde.



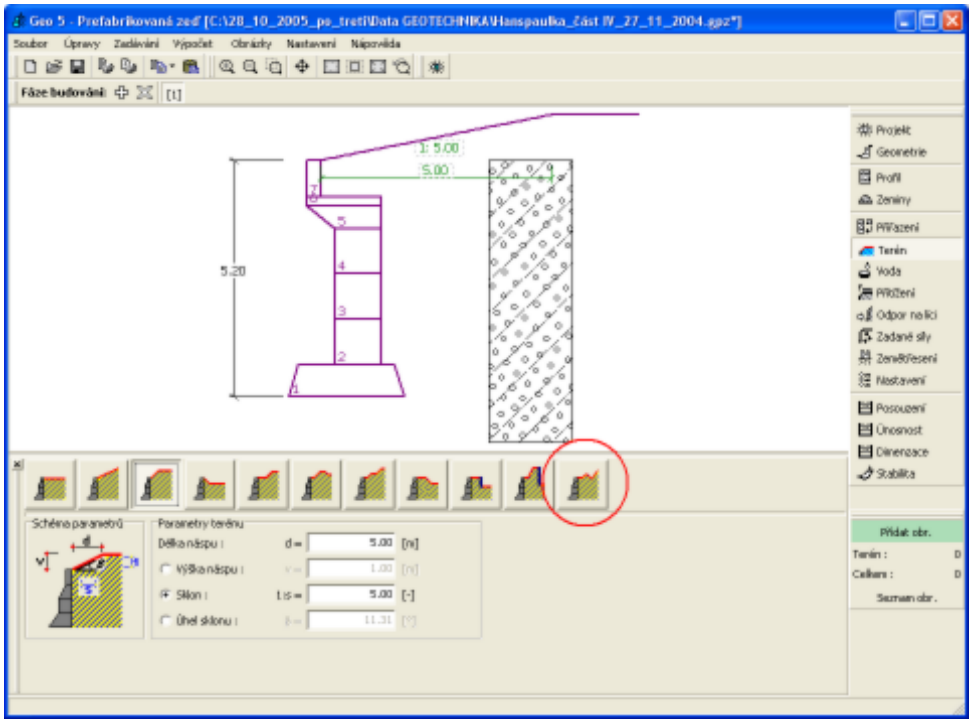
Rám "Přiřazení"

Terén

V rámu "Terén" se stiskem tlačítka volí tvar terénu. Zvolený tvar s grafickou nápovědou ("Schéma parametrů") zadávaných hodnot se zobrazuje v levé části rámu. Tvar terénu lze editovat buď v rámu vložením hodnot do zadávacích polí, nebo na desktopu pomocí aktivních kót.

Poslední volbou v nabídce je obecný tvar terénu. V tomto případě rám obsahuje tabulku se seznamem bodů terénu. První bod terénu se souřadnicí [0,0] leží na horním bodu konstrukce.

Výpočet zemních tlaků při šikmém terénu je popsán v teoretické části nápovědy "Průběh zemních tlaků při členitém terénu".



Rám "Terén"

Voda

V rámu "Voda" se stiskem tlačítka volí typ vody. Zvolený typ s grafickou nápovědou ("Schéma parametrů") zadávaných hodnot se zobrazuje v levé části rámu. Parametry vody (h1, h2...) lze editovat buď v rámu vložením hodnot do zadávacích polí, nebo na desktopu pomocí aktivních kót.

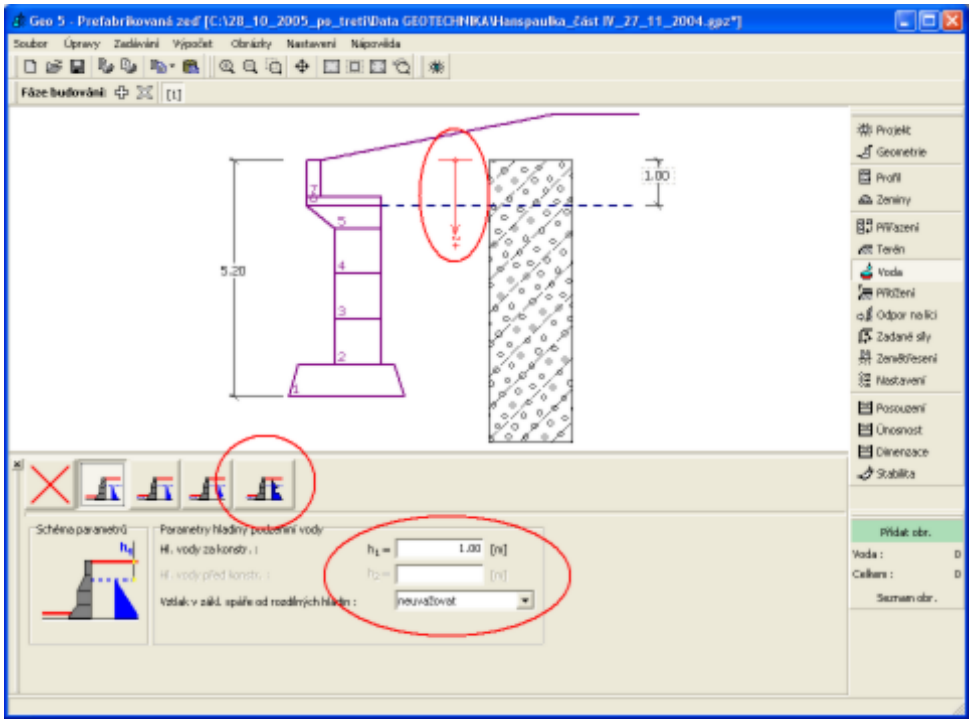
V rozbalitelném seznamu se zadává, zda se předpokládá působení vztlaku vody od rozdílných hladin v základové spáře. Vztlak může být lineární, parabolický nebo není vůbec uvažován. Vztlak v základové spáře od rozdílných hladin je v posouzení zdi zaveden zvláštní silou.

Poslední volbou je ruční zadání pórového tlaku před i za konstrukcí. V rámu se zobrazí dvě záložky "Před konstrukcí" a "Za konstrukcí" s tabulkami. Do tabulek se zadávají hodnoty pórového tlaku před, nebo za konstrukcí v hloubce "*z*" (viz osa *z*)

Hladinu vody lze zadat i nad konstrukcí resp. nad zemní profil - v tomto případě zadáme hloubku vody zápornou hodnotou.

Výpočet zemních tlaků s vlivem vody je popsán v teoretické části nápovědy "Vliv vody".

Dále program umožňuje zadat hloubku tahových trhlin vyplněných vodou.



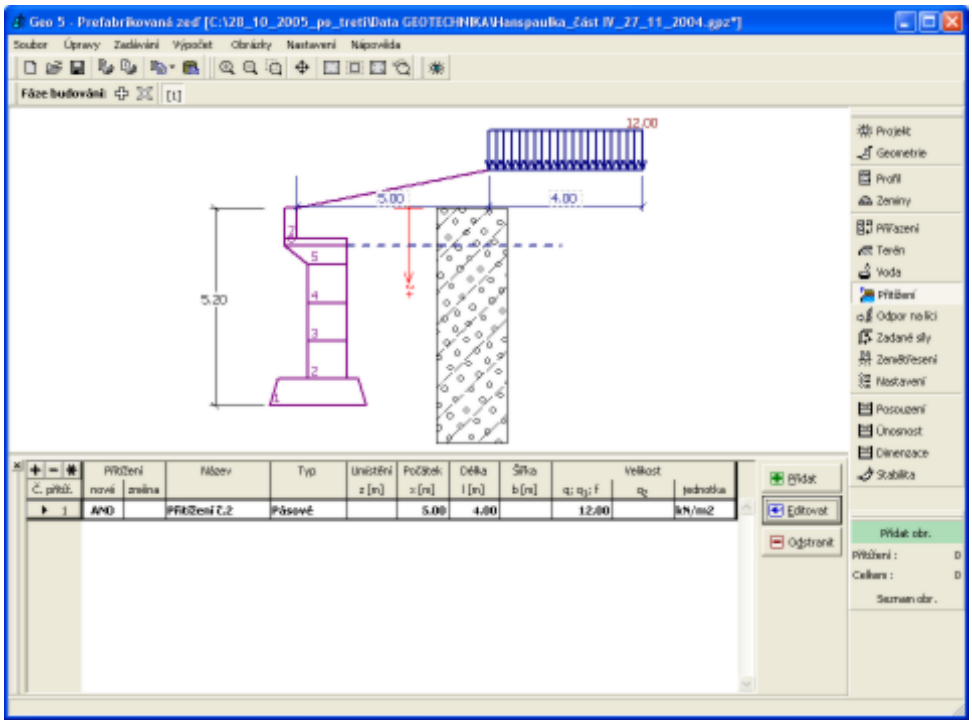
Rám "Voda"

Přitížení

Rám "Přitížení" obsahuje tabulku se seznamem zadaných přitížení. Přidání (editace) přitížení se provádí v dialogovém okně "Nové (editace) přitížení". Zadaná přitížení lze také editovat na desktopu pomocí aktivních kót, resp. aktivních objektů.

Při zadání přitížení v hloubce se zadává z-ová souřadnice od horního bodu konstrukce, kladný směr do hloubky. V případě že přitížení je zadáno mimo terén, program při výpočtu ohlásí chybu.

Výpočet zemních tlaků od přitížení je popsán v teoretické části nápovědy "Vliv přitížení"



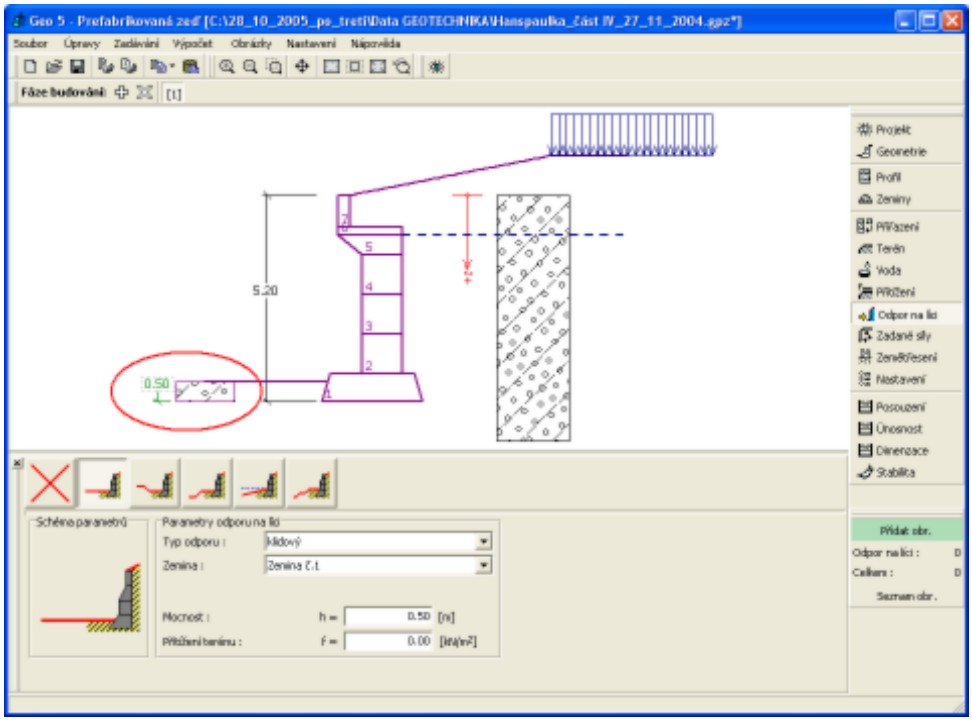
Rám "Přitížení"

Odpor na líci

V rámu "Odpor na líci" se stiskem tlačítka volí tvar terénu a parametry odporu na líci zdi. Zvolený tvar s grafickou nápovědou ("Schéma parametrů") zadávaných hodnot se zobrazuje v levé části rámu. Tvar terénu lze editovat buď v rámu vložením hodnot do zadávacích polí, nebo na desktopu pomocí aktivních kót.

V rámu lze v rozbalitelných seznamech vybrat typ odporu a zeminu (rozbalitelný seznam obsahuje zeminy zadané v režimu "Zeminy"). V rámu lze také zadat velikost přitížení terénu před zdí, popř. mocnost zeminy nad nejnižším bodem konstrukce zdi.

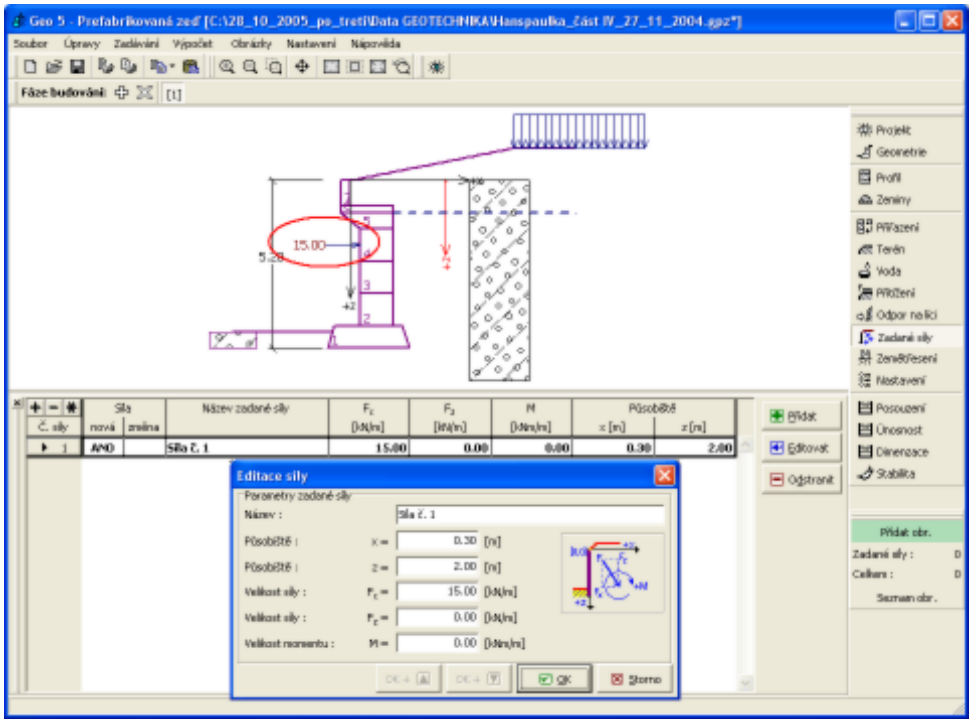
Odpor na líci konstrukce může být zadán jako klidový tlak, pasivní tlak, nebo snížený pasivní tlak. Výsledná síla od sníženého pasivního tlaku se počítá jako výslednice sil od tlaku v klidu a pasivního tlaku vynásobených příslušným součinitelem podle zadání typu sníženého pasivního tlaku.



Rám "Odpor na líci"

Zadané síly

Rám "Zadané síly" obsahuje tabulku se seznamem zadaných sil působících na konstrukci. Přidání (editace) sil se provádí v dialogovém okně "Nová síla (Editace síly)". Zadané síly lze také editovat na desktopu pomocí aktivních objektů.

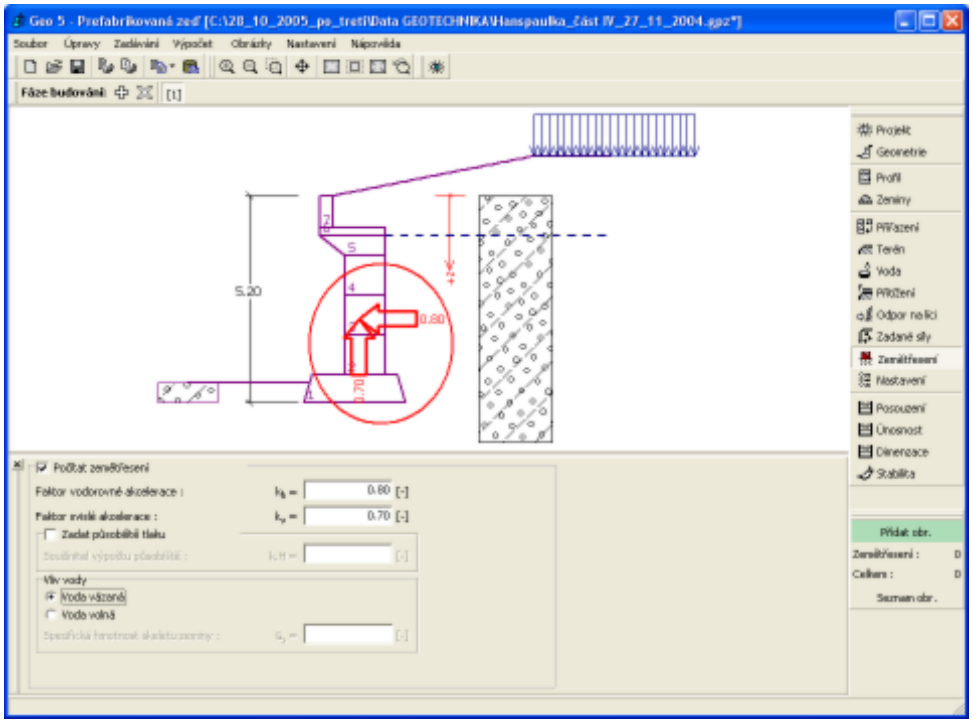


Rám "Zadané síly"

Zemětřesení

V rámu "Zemětřesení" se zadávají parametry zemětřesení. Směry zadaných účinků zemětřesení jsou zobrazeny na desktopu.

Výpočet zemních tlaků s vlivem zemětřesení je popsán v teoretické části nápovědy "Vliv zemětřesení".



Rám "Zemětřesení"

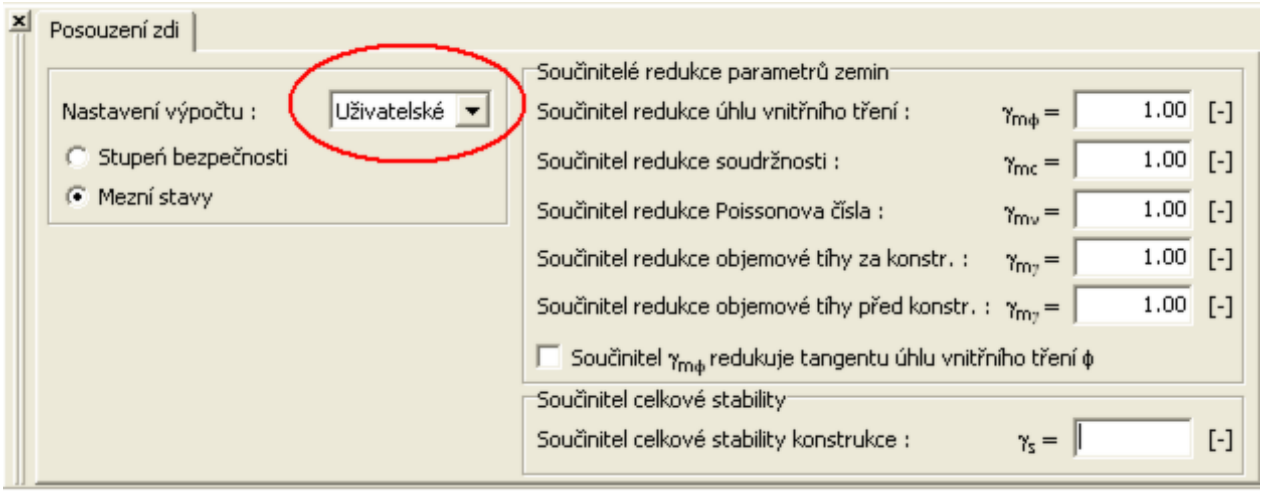
Nastavení

Rám "Nastavení" obsahuje základní nastavení pro výpočet zemních tlaků. Program nabízí přednastavení pro různé země (Česká republika, Německo, Francie, ...) a "Standardní

nastavení" doporučené autory programu pro země neuvedené v seznamu. Při změně nastavení v rozbalitelném seznamu se změní hodnoty součinitelů redukce parametrů zemin v příslušných okénkách.

Při posouzení podle teorie mezních stavů se zadává také součinitel celkové stability konstrukce. Tímto součinitelem se v celkovém posouzení zdi přenásobuje moment vzdorující M_{vzd} a posouvající síla vzdorující H_{vzd}.

Pro volbu "Uživatelské nastavení" lze zadat libovolné nastavení výpočtu. Jestliže vybereme možnost "Redukce parametrů zemin", lze ve vstupních polích zadávat jednotlivé hodnoty součinitelů redukce parametrů zemin (např. doporučené hodnoty součinitelů podle EC7).


Posouzení zdi
Nastavení výpočtu :
Uživatelské
Stupeň bezpečnosti
Mezní stavy
Součinitelé redukce parametrů zemin
Součinitel redukce úhlu vnitřního tření : γmφ = 1.00 [-]
Součinitel redukce soudržnosti : γmc = 1.00 [-]
Součinitel redukce Poissonova čísla : γmν = 1.00 [-]
Součinitel redukce objemové tíhy za konstr. : γmγ = 1.00 [-]
Součinitel redukce objemové tíhy před konstr. : γmγ = 1.00 [-]
Součinitel γmφ redukuje tangentu úhlu vnitřního tření φ
Součinitel celkové stability
Součinitel celkové stability konstrukce : γs = [-]

Rám "Nastavení"

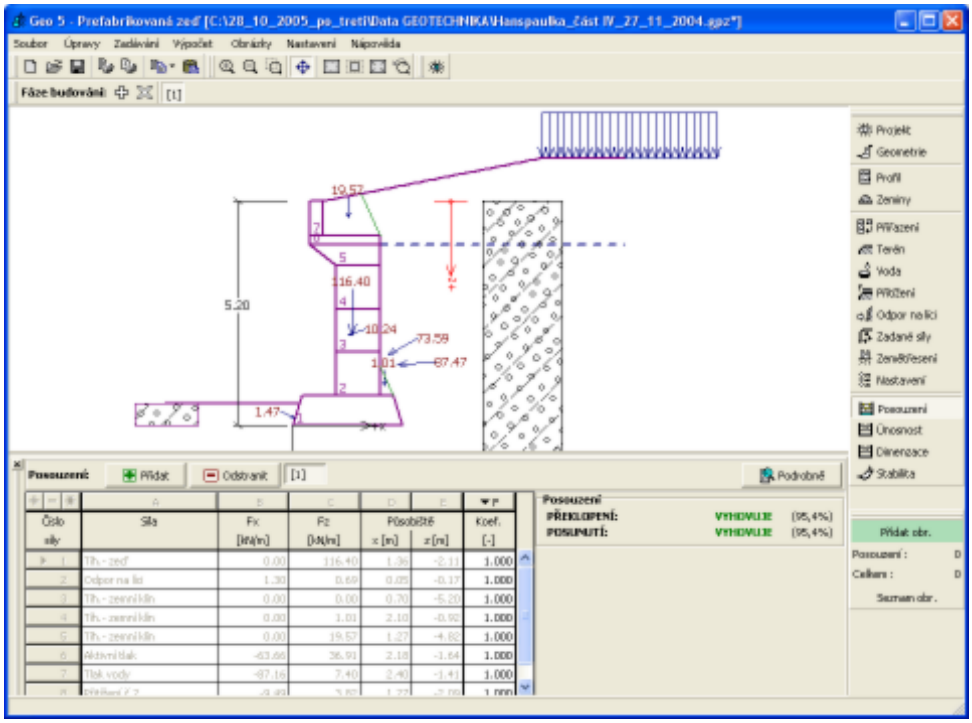
Posouzení

V rámu "Posouzení" jsou zobrazeny výsledky výpočtu. Pro jednu zadanou úlohu lze provést více výpočtů a to s různými výpočtovými koeficienty výsledných silových účinků.

Postup vlastního posouzení zdi je popsán v teoretické části nápovědy.

Spočtené síly jsou zobrazeny na desktopu a aktualizují se okamžitě při jakékoliv změně dat a nastavení v rámu. V pravé části rámu se zobrazuje výsledek posouzení zdi na překlopení a posunutí. Tlačítko "Podrobně" otevírá dialogové okno, které obsahuje detailní výpis výsledků posouzení.

Zobrazení kresby lze měnit v dialogovém okně "Nastavení stylu zobrazení".



Rám "Posouzení"

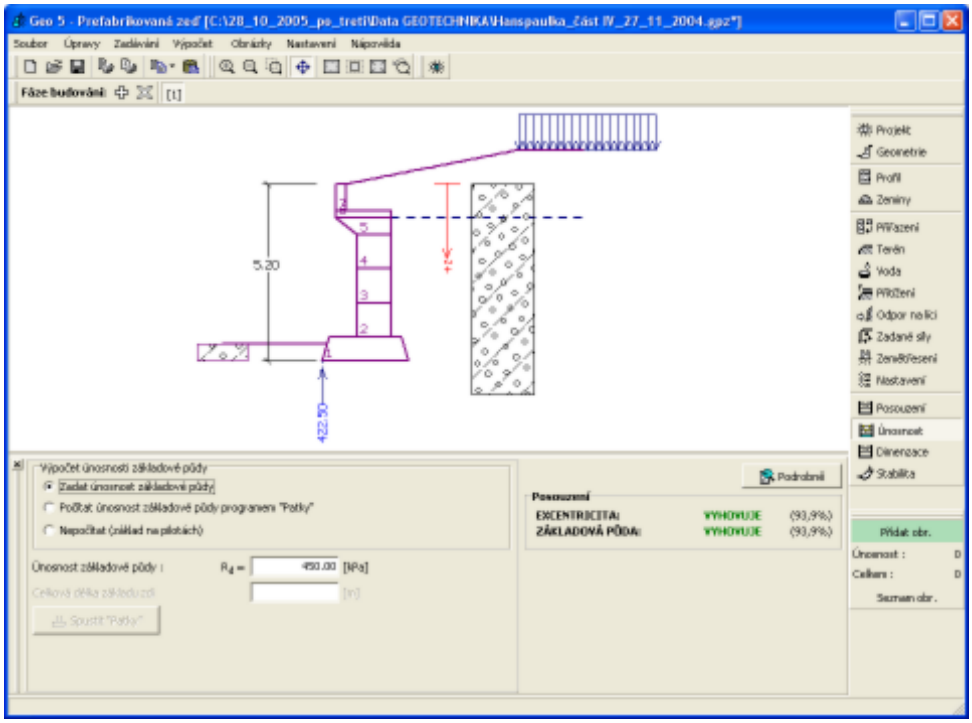
Únosnost

V rámu "Únosnost" jsou zobrazeny výsledky výpočtu únosnosti základové půdy. Napětí v základové spáře (předpokládá se konstantní) je spočteno ze všech posouzení provedených v rámu "Posouzení". Do programu "Patky" se jednotlivá posouzení převezmou jako zatěžovací stavy.

Rám obsahuje tři základní volby výpočtu:

- Zadat únosnost základové půdy
  V zadávacím poli se zadává únosnost základové půdy. Výsledky posouzení zdi na excentricitu a únosnost základové půdy se zobrazují v pravé části rámu. Tlačítko "Podrobně" otevírá dialogové okno, které obsahuje detailní výpis výsledků posouzení únosnosti základové půdy.
- Počítat únosnost základové půdy programem "Patky"
  Pomocí tlačítka "Spustit "Patky" se spustí program " Patky", ve kterém lze spočítat únosnost základové půdy, popř. sednutí a natočení základu. Po provedení výpočtů odejdeme tlačítkem "OK" - výsledky resp. zadané obrázky se přenesou do protokolu u výpočtu v programu "Prefa zeď".
  Pokud není program "Patky" nainstalován, není tlačítko dostupné.
- Nepočítat (základ na pilotách)
  Únosnost základové půdy se nepočítá.

Zobrazení kresby lze měnit v dialogovém okně "Nastavení stylu zobrazení".



Rám "Únosnost"

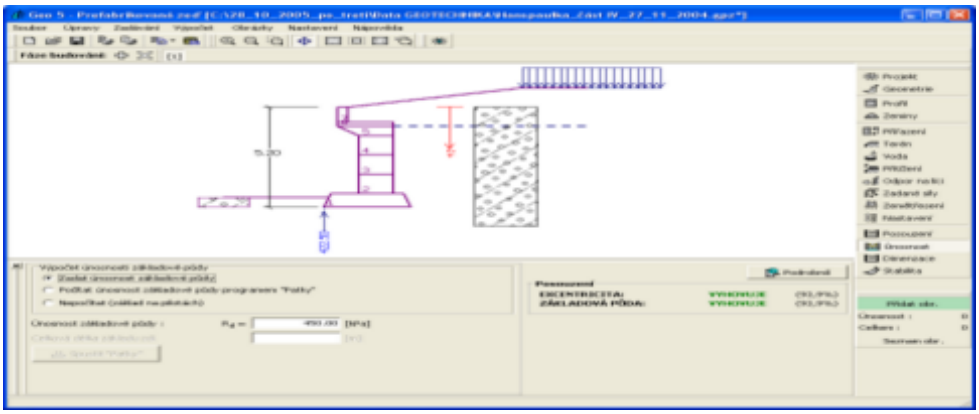
Dimenzace

V rámu "Dimenzace" se posouzení jednotlivé řady svahovek. Pro jednu zadanou úlohu lze provést více výpočtů a to s různými výpočtovými koeficienty výsledných silových účinků.

V rámu lze volit posouzení konkrétních řady svahovek, nebo automatické posouzení. V módu automatického posouzení program najde řadu svahovek s největším využitím a zobrazí ji na obrazovce.

Posouzení řad svahovek je provedeno v závislosti na zadání v rámu "Nastavení" podle mezních stavů nebo stupně bezpečnosti.

Je-li konstrukce tvořena lavičkami, není únosnost na rozdrcení pod lavičkou počítána – konstrukci je nutné posoudit nadvakrát – podle obrázku.



Program Abutment

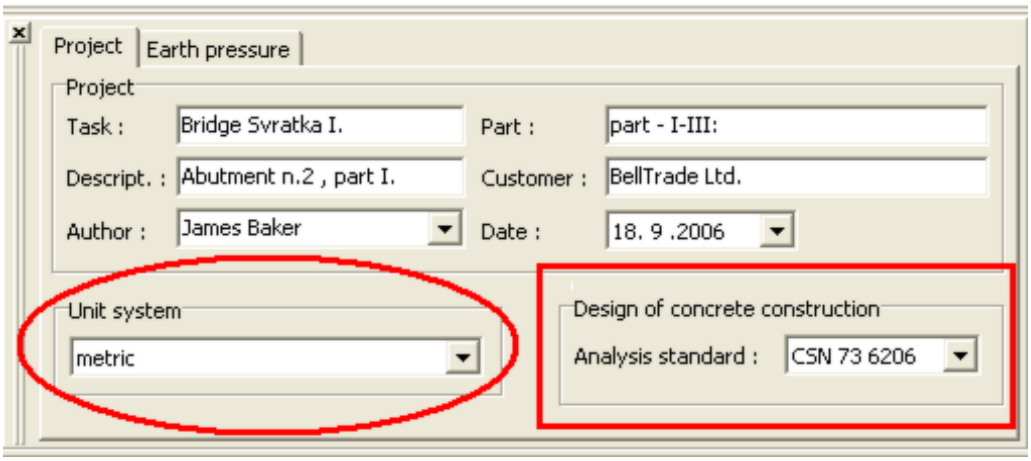
Project

The "Project" frame is used to input the basic project data and to specify the overall setting of the analysis run. The "Project" tab sheet contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc.

The "Project" tab sheet also allows the user to switch analysis units (metric / imperial).

The "Project" tab sheet further serves to specify a standard for concrete structures dimensioning. Referring to the selected standard the types of concrete and steel are then inputted in the"Material" frame. The dimensioning of cross-sections of the analyzed structure is then performed in the "Dimensioning" frame. Only ACI standard is available when selecting imperial units. While selecting standard it is also necessary to specify whether the stress in the foundation joint used for verification of a wall front key is assumed uniform (CSN) or trapezoidal (Eurocode).

The "Earth pressures" tab sheet serves to choose the basic theory or standards to be followed in the solution of a given problem.



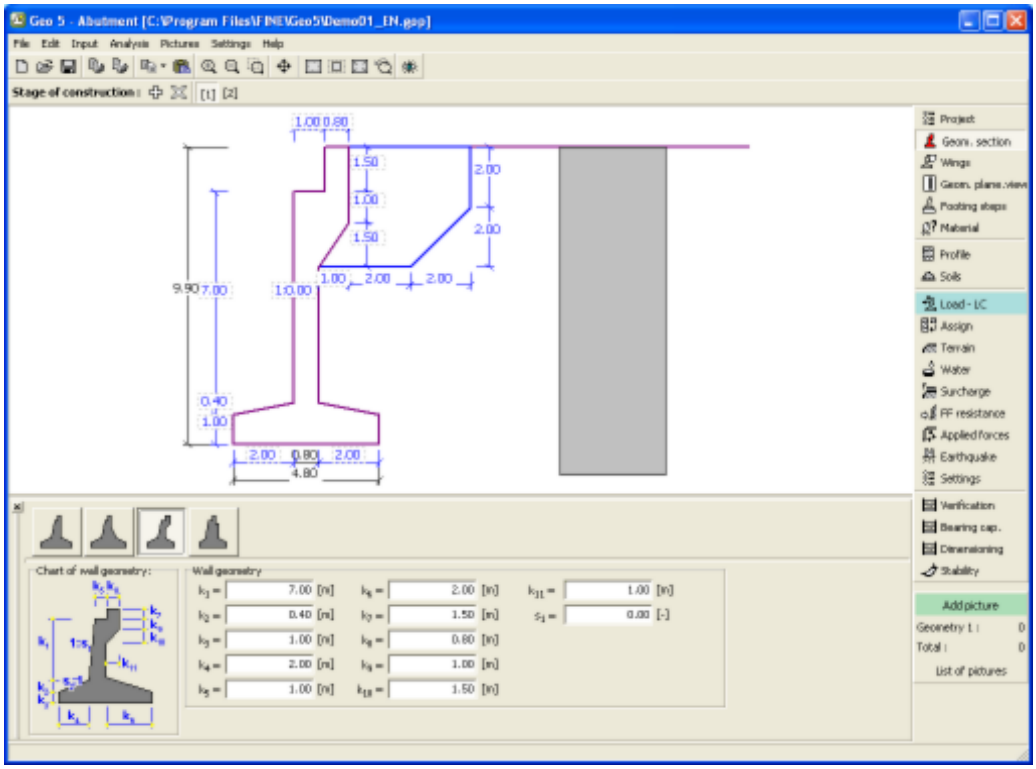
Frame "Project" - tab sheet "Project"

Geometry cut

The frame "Geom. cut" allows for selecting the shape of bridge abutment. The selected shape with a graphic hint "Wall geometry chart" appears in the left part of the frame. The shape of a wall can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

In case the structure is composed of inclined segments it is required to enter the ratio of sides of an inclined segment 1:x. The straight structure is specified by entering the value zero.

The frame serves to specify the final shape of abutment including the closure wall. The abutment can be verified also for the construction state (without the closure wall) based on the choice in the frame "Loading - LC". The abutment length and the length of abutment foundation is specified in the frame "Geometry plane view ".



Frame "Geometry cut"

Wings

The frame "Wings" allows for inputting the bridge wings dimensions. The wings can be either symmetrical or unsymmetrical. Assuming unsymmetrical wings requires inputting the right and left wing dimensions separately. The screen always displays the currently inputted wing – only the left wing is then visualized in the remaining frames.

The frame "Geometry plane view" can also be used to input or edit the wing thicknesses and lengths.

The Wing-abutment joint cross-section can also be verified in the frame "Dimensioning". The loading due to moment is considered. The whole wing is loaded by active pressure developed behind the wall. The "Dimensioning" dialogue window serves to input the magnitude of surface surcharge to determine the wing pressure. The surcharge specified in the frame "Surcharge" is then not taken into account and the terrain behind the wing is considered as flat. The resulting moment applied to the joint is obtained by multiplying the overall magnitude of soil pressure acting on the wall surface and by the difference of centroids of the pressure resultant and the joint.

The length of cross-section used for dimensioning is considered by default as the wing height – a different length of wing-abutment joint can also be specified after selecting the option "Reduce for dimensioning".

When using prolonged wing walls it is possible to input dimensions of the foundation below the wall. Such foundation jumps are reflected in the analysis by computing a fictitious width of the foundation as:

$$d_{fict} = \frac{A_{tot}}{S}$$

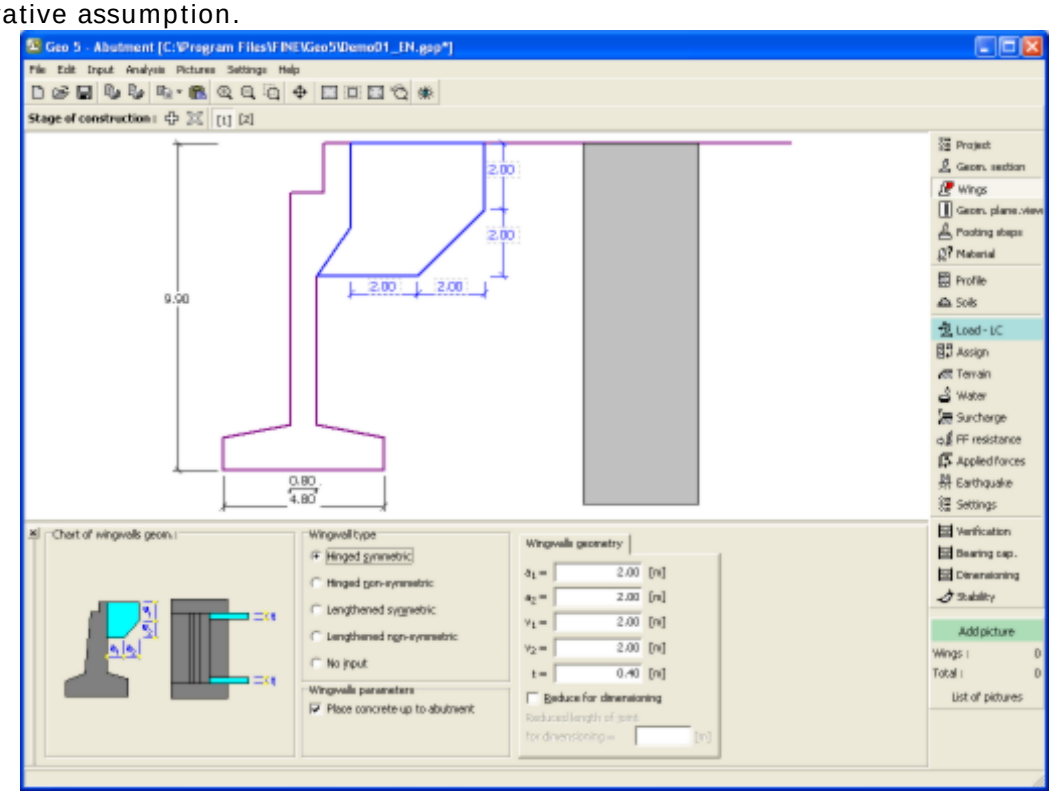
where: A_{tot} - overall area of foundation including all jumps

S - length of abutment foundation

d_{fict} - fictitious width of foundation for verification analysis

The foundation is then considered as being rectangular, which is simplified but rather

conservative assumption.

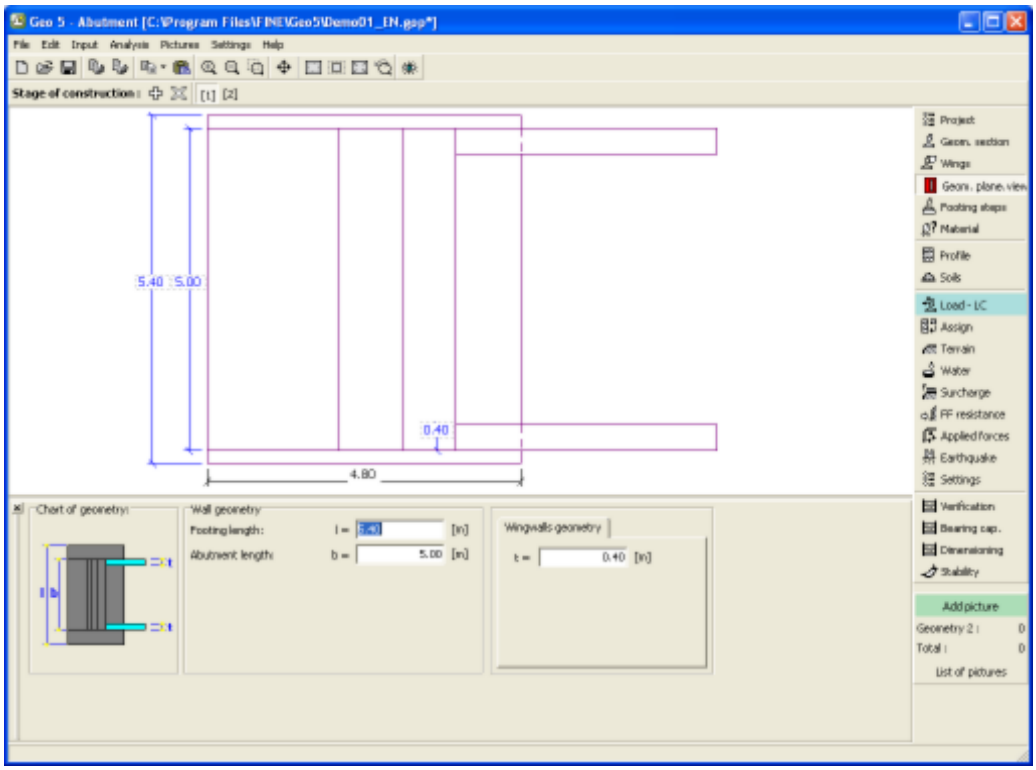


Frame "Wings"

Geometry plane view

The frame "Geometry plane view" allows for inputting the abutment length, length of abutment foundation and also dimensions of abutment wings.
Dimensions can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

For details on the effect of abutment dimensions on verification analysis we refer the reader to section "Calculating of abutment forces".

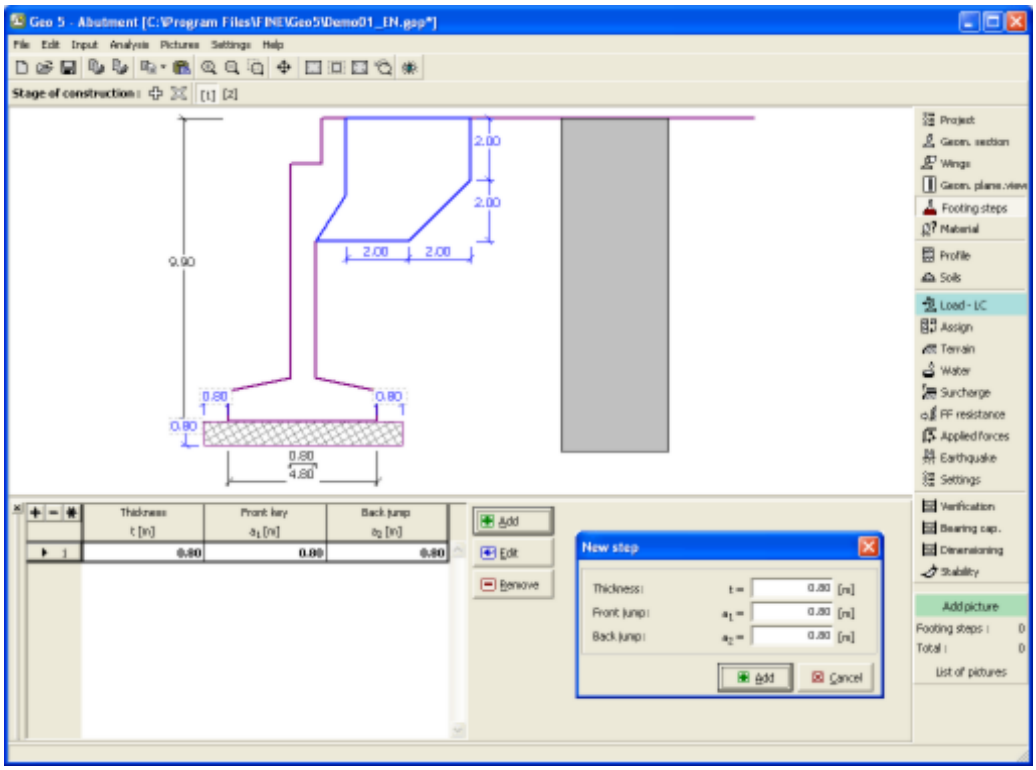


Frame "Geometry plane view"

Foundation steps

The frame "Foundation steps" serves to input the steps of foundation below abutment. This option thus allows for specifying additional shapes of bridge abutment.

Adding (editing) foundation step is performed in the "New step" dialogue window. Inputted foundation steps can be edited on the desktop with the help of active dimensions or active objects, respectively.



Frame "Foundation steps"

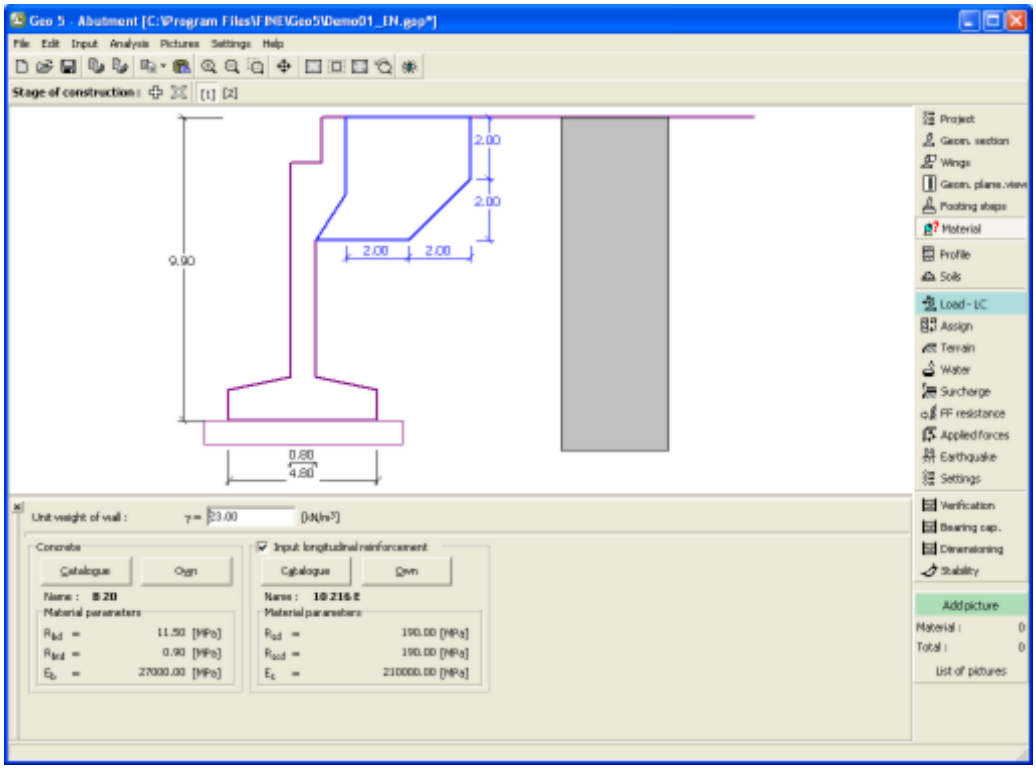
Material

The "Material" frame allows for the selection of material parameters for concrete and longitudinal steel reinforcements.

Two options are available when selecting the material type:

- the "Catalogue" button opens the "Material catalogue" dialogue window (for concrete or steel reinforcements), the list of materials then serves to select the desired material
- the "User" button opens the "Edit material – concrete" dialogue window (for concrete) or the "Edit material – concrete steel" dialogue window (for longitudinal steel reinforcements), which allows for manual specification of material parameters

The catalogues content depends on the selection of standard for the design of concrete structures set in the "Project" frame. The input field in the upper part of the frame serves to specify the abutment unit weight.



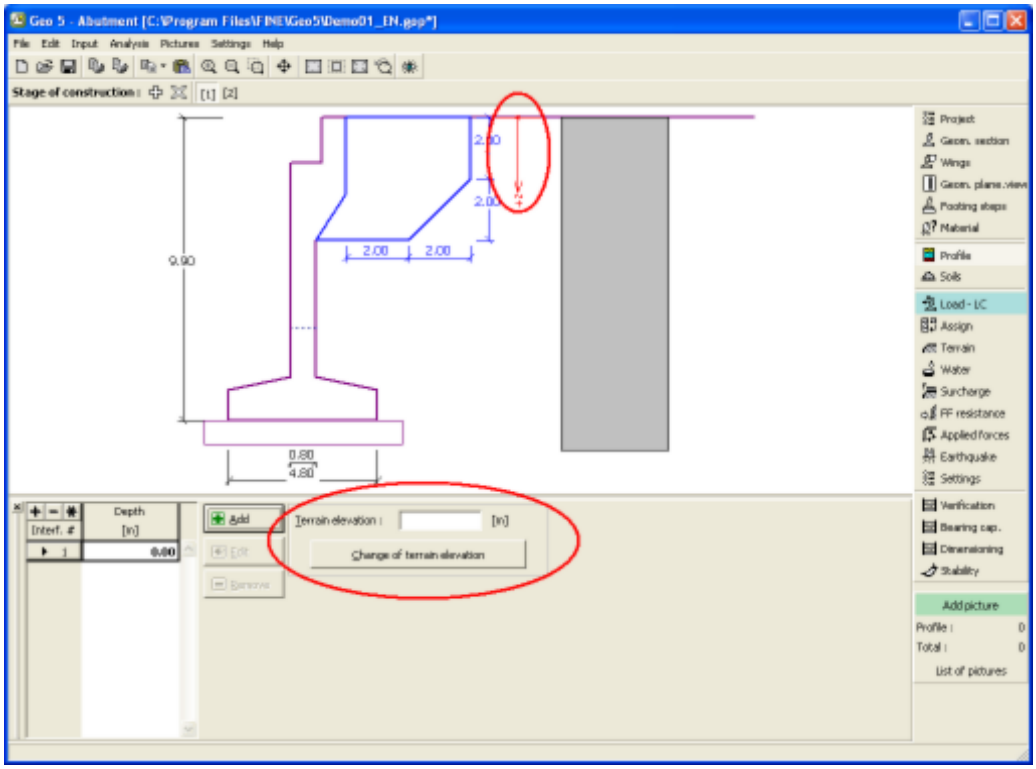
Frame "Material"

Profile

The "Profile" frame contains a table with a list of inputted interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "Add (edit) interface" dialogue window. The z-coordinate measured from the top point of a structure is specified.

The program allows for raising or lowering the top point of a structure in the "Change terrain elevation" dialogue window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".



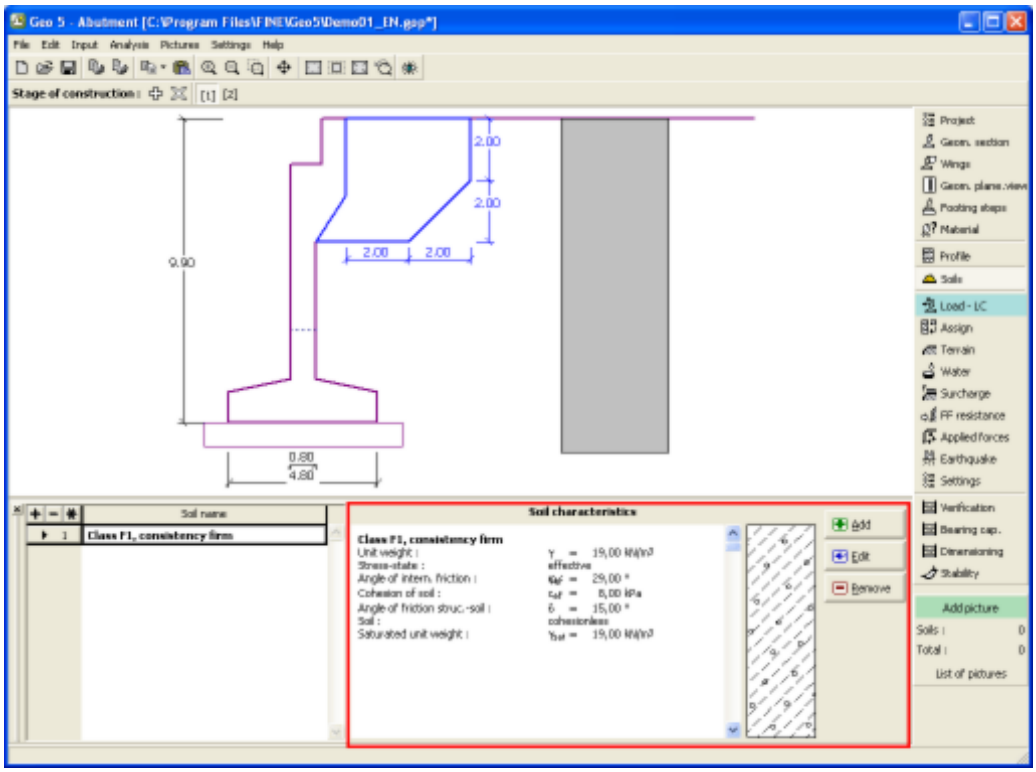
Frame "Profile"

Soils

The "Soils" frame contains a table with a list of inputted soils. The table also provides information about currently selected soil displayed in the right part of the frame.

Adding (editing) a soil is performed in the "Add (edit) soil" dialogue window.

The soil characteristics are specified in the program "Cantilever wall". These characteristics are further specified in chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".



Frame *"Soils"*

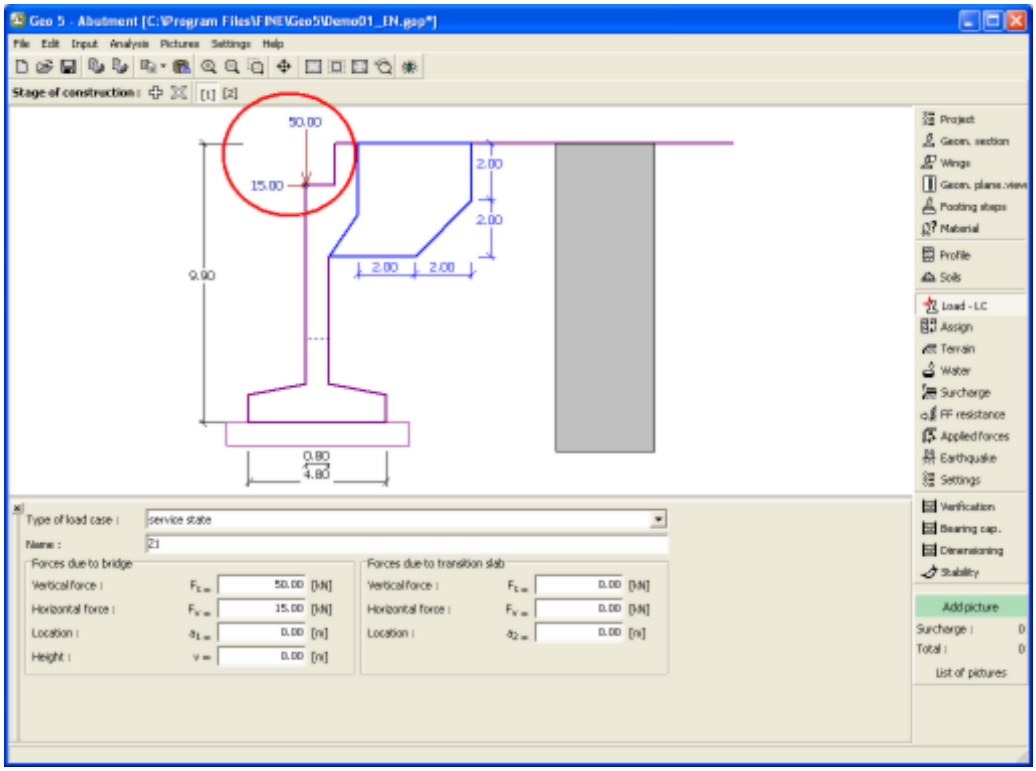
Loading - LC

The frame "Loading – LC" serves to specify individual loading cases (construction, service) and the loading caused by the bridge and approach slab. Verification and dimensioning analyses of the whole bridge abutment or only its part are carried out according to the specified type of LC.

No load specified in the case of construction state and the abutment is verified in a given stage of construction without a closure wall and bridge wings.

In the case of service state the abutment is loaded by the bridge and approach slab, the whole abutment is verified.

For abutment verification it appears advantageous to exploit the stage of construction and specify in individual stages different load cases (e.g. construction state, service state without live load, service state with all loads). Individual stages then allow inputting different loads, surcharges, terrain shapes, type of pressure analysis (active, at rest), design coefficients, etc.

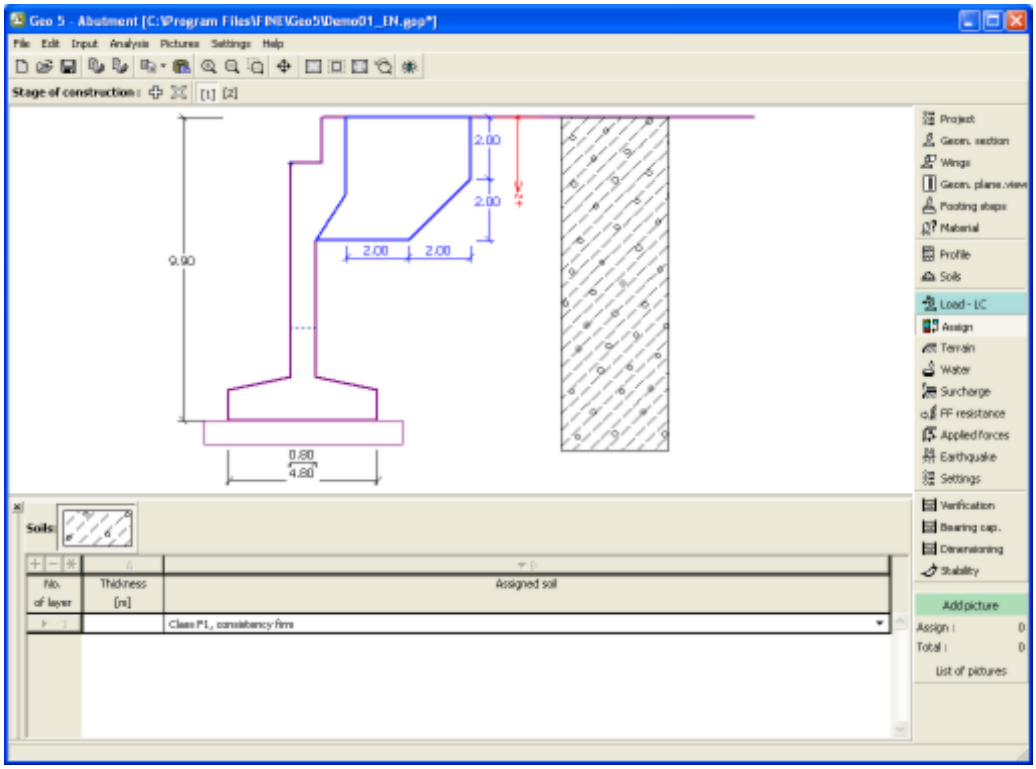


Frame "Loading - LC"

Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

Procedure to assign soil into a layer is described in details herein.



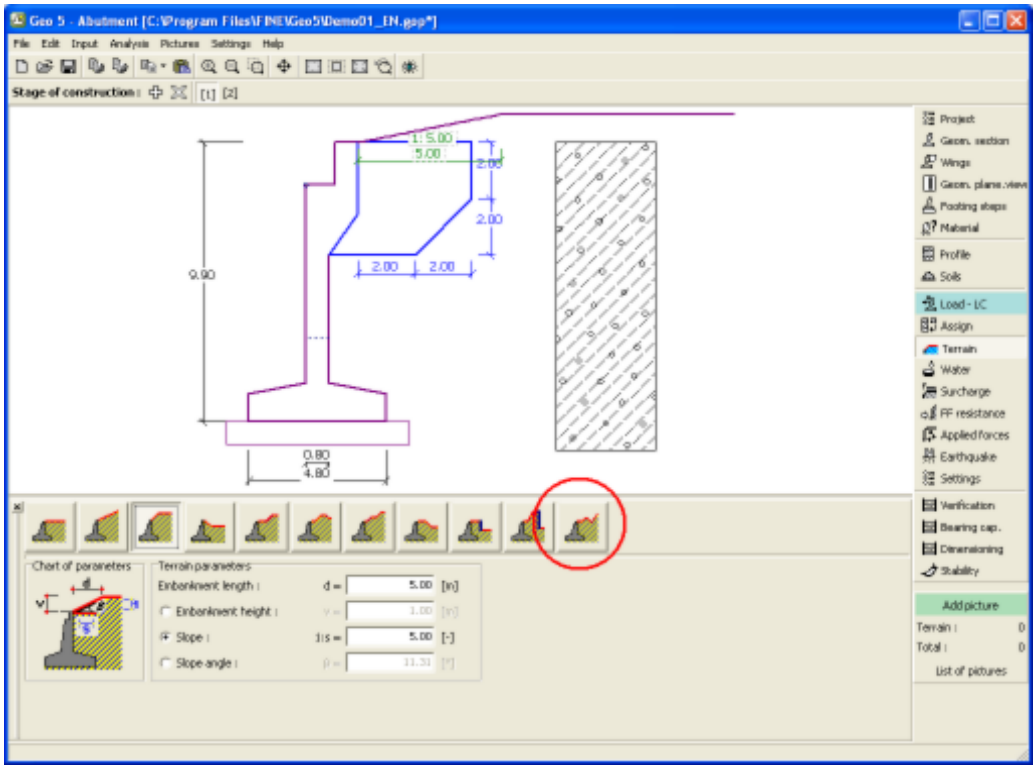
Frame "Assign"

Terrain

The "Terrain" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0, 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the hint, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

The "Water" allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. Water parameters (*h1, h2*...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

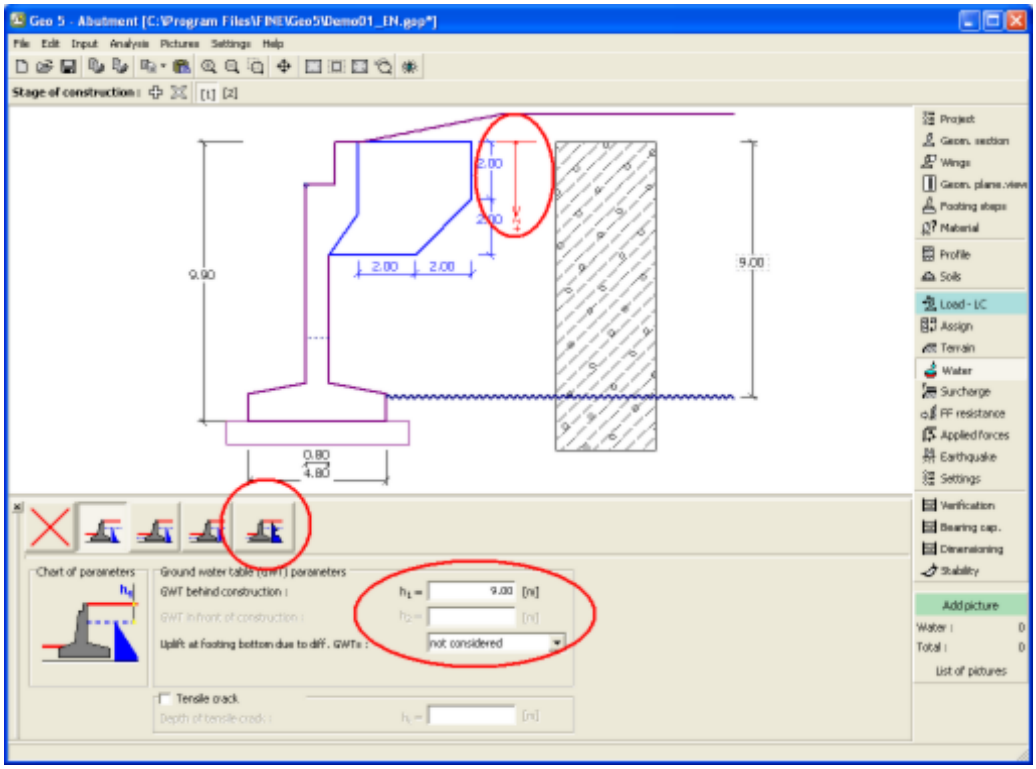
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic or it may not be considered at all. When verifying the wall, the uplift pressure in base of footing joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tab sheets "In front of structure" and "Behind structure" appear with tables. The table is filled with values of pore pressure in front, or behind the structure at a depth of "*z*" (z-axis).

The ground water table can also be specified above the structure or earth profile, respectively – in such a case the depth of water is inputted with negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the hint chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



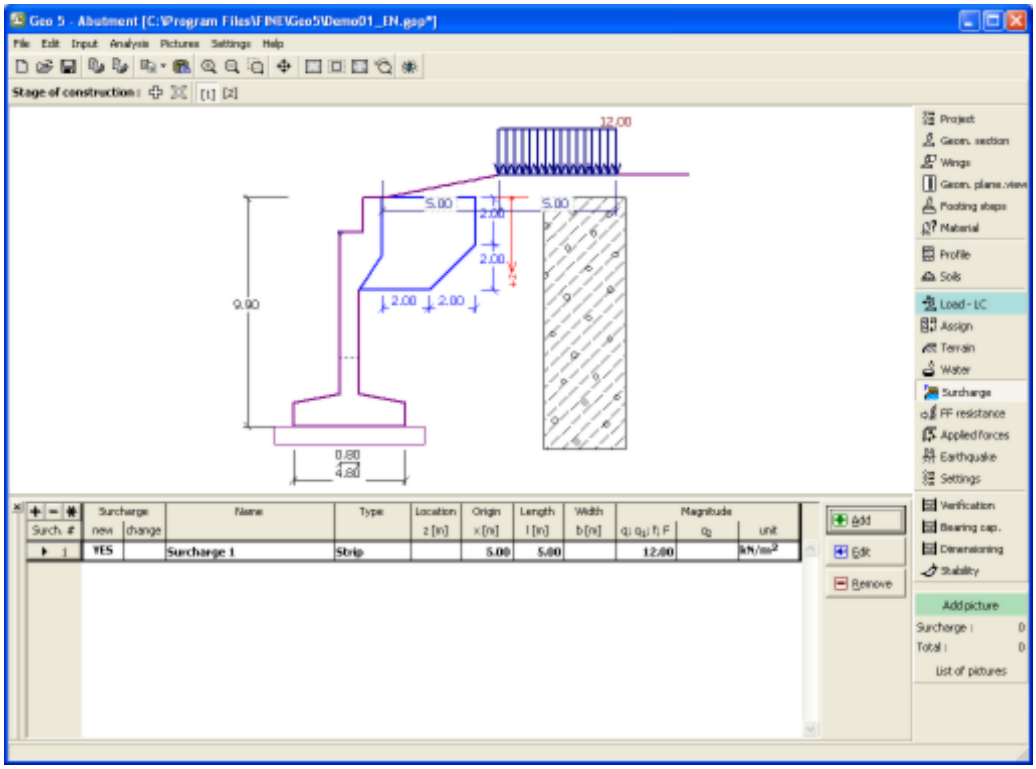
Frame "Water"

Surcharge

The "Surcharge" frame contains a table with a list of inputted surcharges. Adding (editing) surcharge is performed in the "New (edit) surcharge" dialogue window. The inputted surchages can be edited on the desktop with the help of active dimensions or active objects, respectively.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. Providing the surcharge is found off the terrain the computer prompts an error message.

Analysis of earth pressures due to surcharges is described in the theoretical part of the hint, chapter "Influence of surcharge".



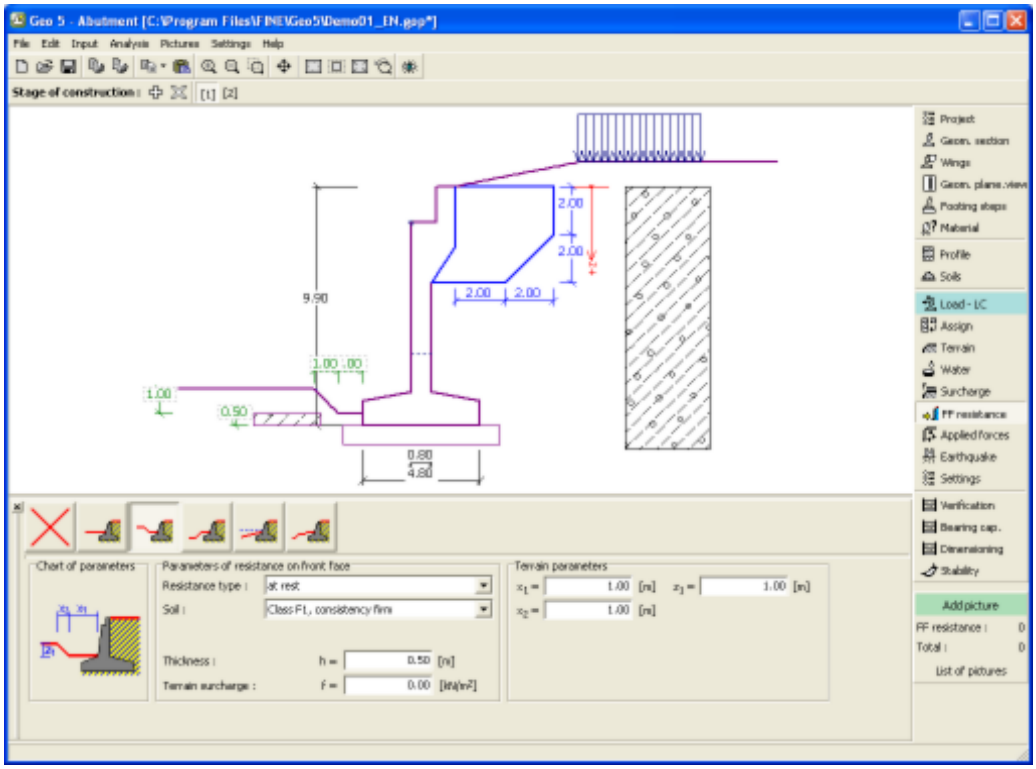
Frame "Surcharge"

Front face resistance

The "Front face resistance" frame allows by pressing the button for specifying the terrain shape and parameters of front face resistance. The selected shape with a graphic hint (" Parameter chart") of inputted values are displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Combo lists in the frame allows the user to select the type of resistance and a soil (the c ombo list contains soils introduced in the regime "Soils"). The magnitude of terrain surcharge in front of the wall or soil thickness above the wall lowest points can also be specified in the frame.

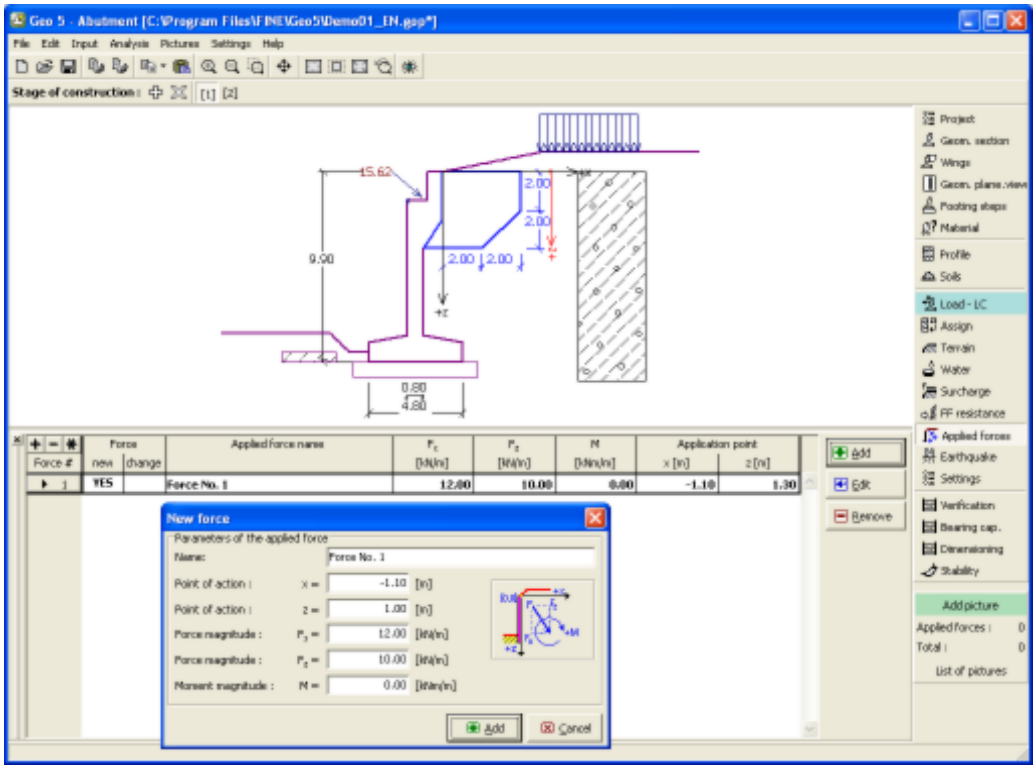
The resistance on a structure front face can be specified as a pressure at rest, passive pressure or reduced passive pressure. The resulting force due to reduced passive pressure is found as a resultant force caused by passive pressure multiplied by a corresponding coefficient, which follows from the inputted type of reduced passive pressure.



Frame "Front face resistance"

Inputted forces

The "Inputted forces" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "New force (edit force)" dialogue window. The inputted forces can also be edited on the desktop with the help of active objects.

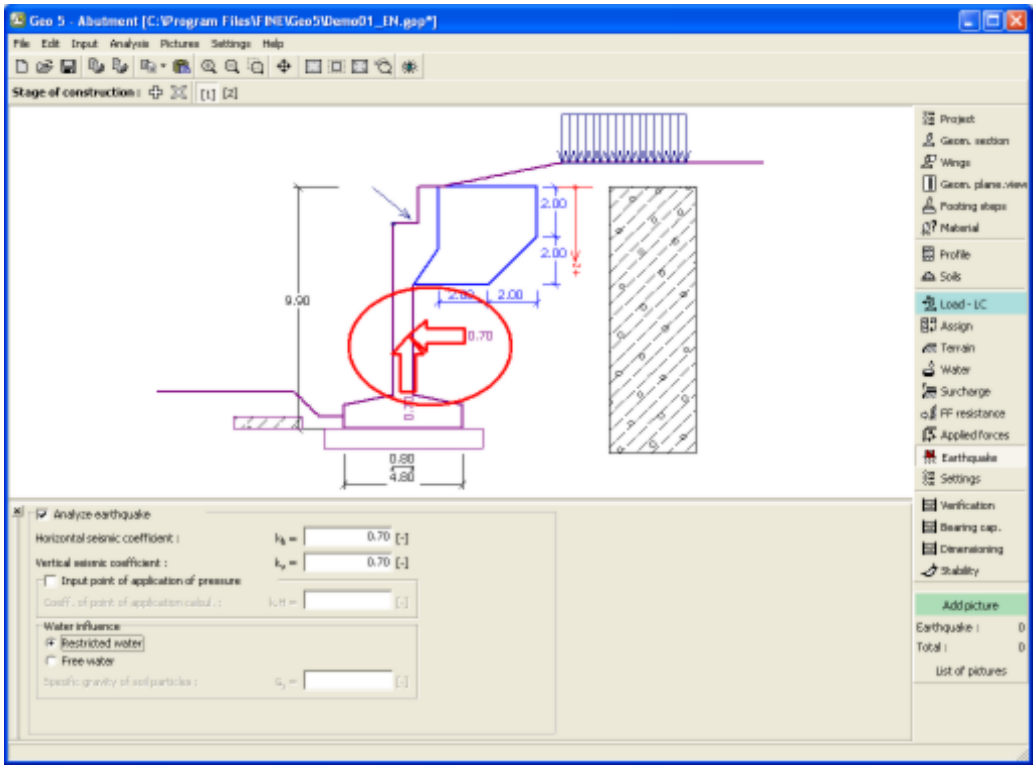


Frame "Inputted forces"

Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of inputted earthquake effects are displayed on the desktop.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the hint in chapter "Influence earthquake".



Frame "Earthquake"

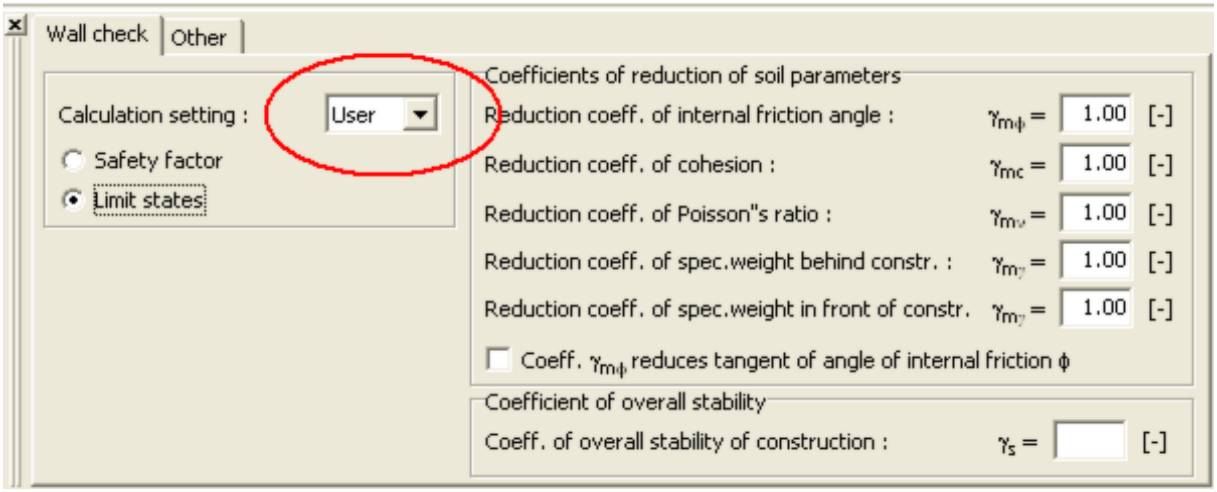
Setting

The "Setting" frame contains basic settings for the analysis of earth pressures. The program offers pre-setting for different countries (Czech Republic, Germany, France,...) and " Standard setting" recommended by the authors of the program for countries no included in the list. While changing settings in the combo list the values of coefficients of reduction of soil parameters in corresponding windows are changed.

Evaluating the structure according to theory of limit states also calls for the input of coefficient of overall stability of a structure. When subjecting the wall to overall verification this coefficient is used to multiply the resisting moment M_{res} and the resisting shear force H_{res}.

An arbitrary analysis setting is available with the option "User setting". Selecting the option "Reduce soil parameters" allows for specifying in input fields individual values of the coefficients of reduction of soil parameters (e.g., recommended values according to EC7-1).

The "Other" tab sheet serves to specify the type of pressure acting on a wall based on the allowable wall deformation. Providing the wall is free to move an active pressure is assumed, otherwise, a pressure at rest is used.


Wall check
Other
Calculation setting :
User
Safety factor
Limit states
Coefficients of reduction of soil parameters
Reduction coeff. of internal friction angle : γmϕ = 1.00 [-]
Reduction coeff. of cohesion : γmc = 1.00 [-]
Reduction coeff. of Poisson"s ratio : γmν = 1.00 [-]
Reduction coeff. of spec.weight behind constr. : γmγ = 1.00 [-]
Reduction coeff. of spec.weight in front of constr. γmγ = 1.00 [-]
Coeff. γmϕ reduces tangent of angle of internal friction ϕ
Coefficient of overall stability
Coeff. of overall stability of construction : γs = [-]

Frame "Setting"

Verification

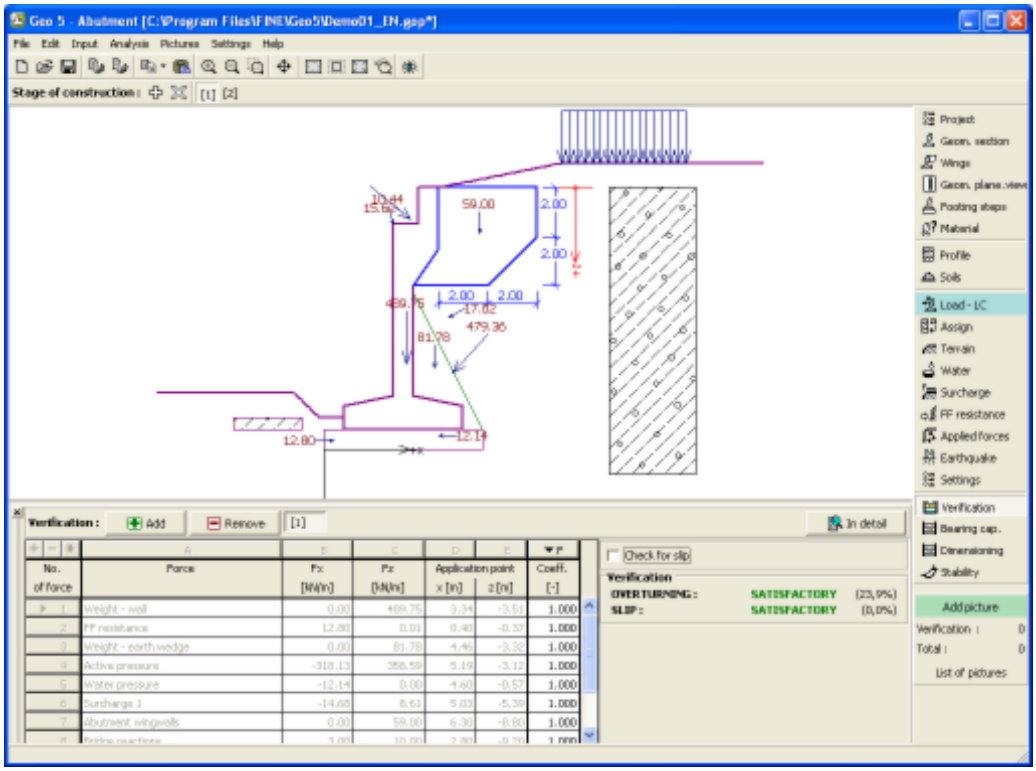
The "Verification" frame shows the analysis results. Several computation with different coefficients of resultant force effects can be carried out for a single task.

The wall is loaded either by active pressure or pressure at rest depending on input in the frame "Setting".

Procedure for wall verification is described in the theoretical part of the hint.

The computed forced are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against overturning and translation. The "In detail" button opens the dialogue window, which contains detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Verification"

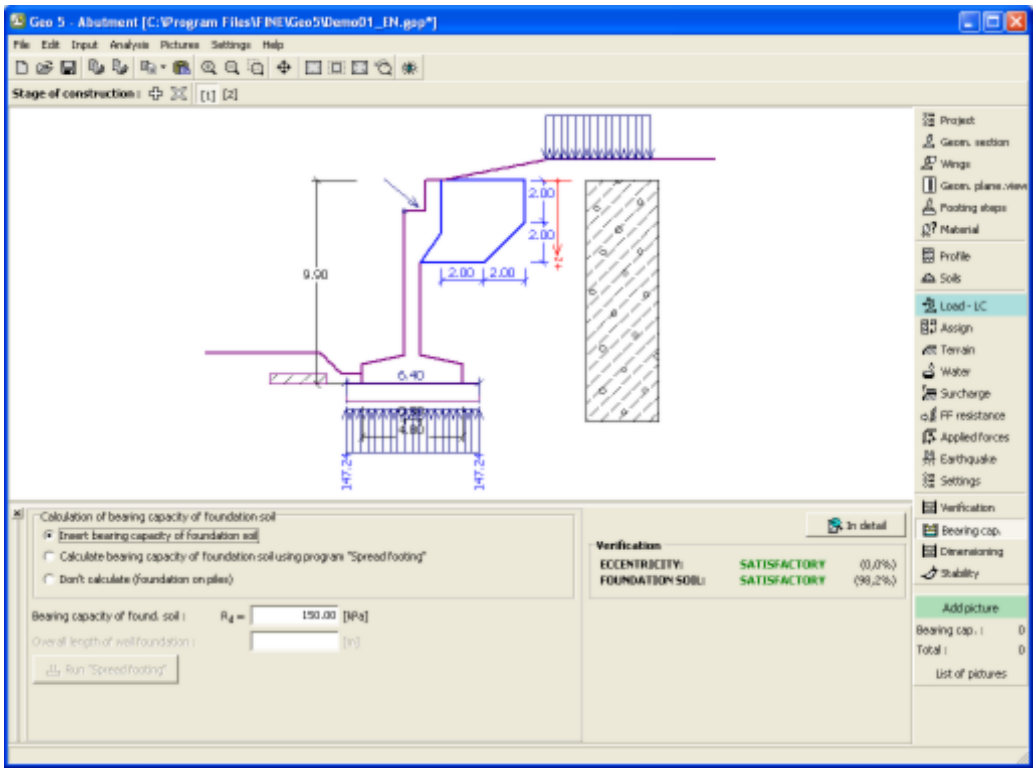
Bearing capacity

The "Bearing capacity" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "Verification". The program "Spread footing" then considers all verifications as loading cases.

Three basic analysis options are available in the frame:

- Input the foundation soil bearing capacity — The input field serves to specify the foundation soil bearing capacity. The results of verification analysis of a soil for eccentricity and bearing capacity are displayed in the right part of the frame. The "In detail" button opens the dialogue window that displays detailed listing of the results of verification analysis of foundation soil bearing capacity.
- Compute the foundation soil bearing capacity using the program "Spread footing" — Pressing the "Run "Spread footing" button starts the program "Spread footing" that allows for computing the soil bearing capacity or settlement and rotation of a footing. Pressing the "OK" button leaves the analysis regime – the results and all plots are copied to the program "Abutment". The program "Spread footing" must be installed for the button to be active.
- Do not compute (pile footing) — The foundation soil bearing capacity is not computed.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Bearing capacity"

Dimensioning

The "Dimensioning" frame serves to design and verify the reinforcement of abutment cross-section – the cross-section subjected to dimensioning is selected in the combo list. The table shows the abutment forces.

Offer of cross-sections that can be verified depands on the selected load case (construction, service). The following cross-sections are available for both the construction and service state:

- Wall stem verification
- Construction joint verification — depth of construction joint from construction top edge is specified
- Wall jump verification — type of assumed stress acting in construction joint for verification (linear, constant) can be selected in the frame "Project"

The service state makes also possible to verify:

- Verification of closure wall
- Verification wing - abutment — the surface surcharge due to terrain is inputted, for actual analysis we refer to section "Wings"

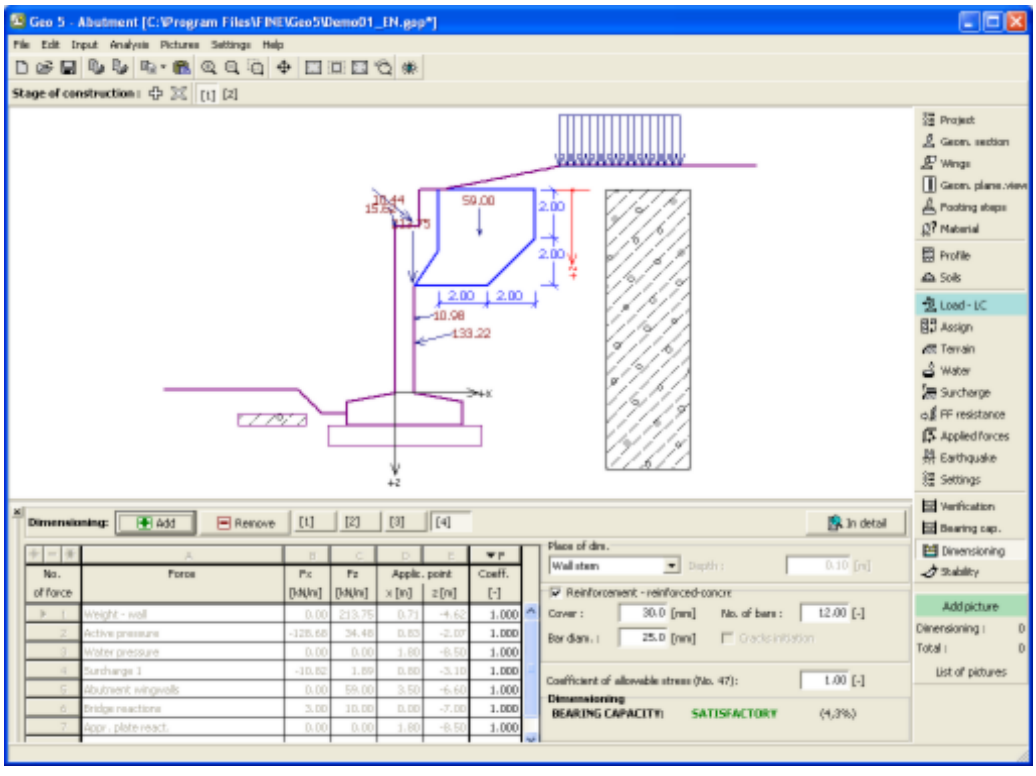
The abutment is loaded either by active pressure or pressure at rest depending on the input specified in the frame "Settting", an active earth pressure is used when analysing wing walls.

Procedure to derive distribution of internal forces in individual cross-sections is described in the theoretical part of this hint.

Dimensioning of the steel-reinforced concrete structure is performed according to the standard set in the frame "Project". Verification analysis based on the standard CSN 73 6206 "Design of concrete and steel reinforced concrete bridge structures" is described herein.

Several computations for various cross-sections can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "In detail" button opens the dialogue window that contains detailed listing of the dimensioning results.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.

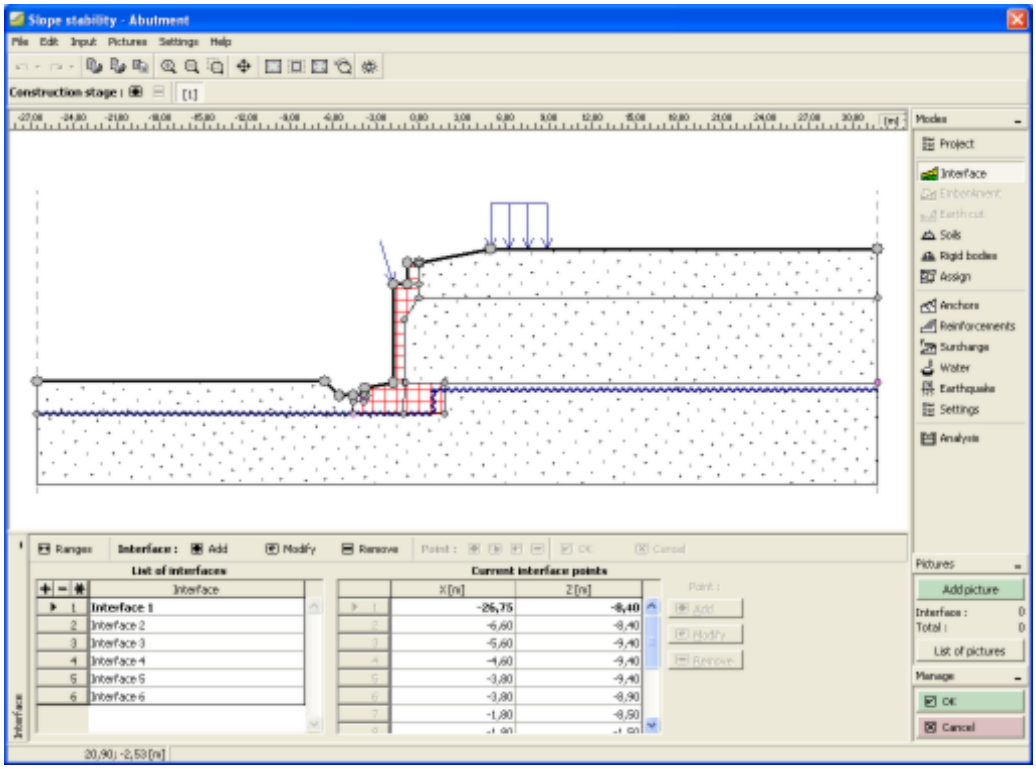


Frame "Dimensioning"

Stability

Pressing the "Stability" button launches the "Slope stability" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "Slope stability" is installed.

After completing all analyses press the "OK" button to leave the program – all data are then carried over to the analysis protocol of the "Abutment" program.



Frame "Stability"

Program Nailed slopes

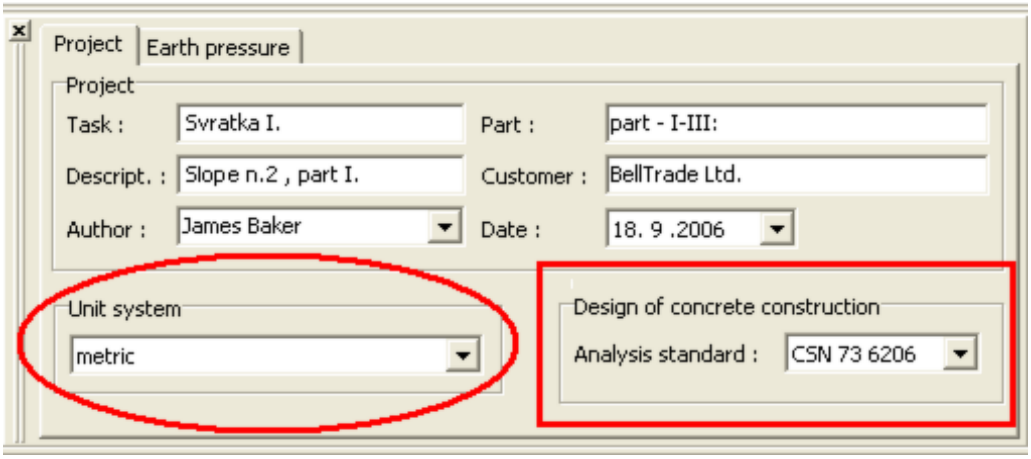
Project

The "Project" frame is used to input the basic project data and to specify the overall setting of the analysis run. The "Project" tab sheet contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc.

The "Project" tab sheet also allows the user to switch analysis units (metric / imperial).

The "Project" tab sheet further serves to specify a standard for concrete structures dimensioning. Referring to the selected standard the types of concrete and steel are then inputted in the "Material" frame. The dimensioning of cross-sections of the analyzed structure is then performed in the "Dimensioning" frame. Only ACI standard is available when selecting imperial units.

The "Earth pressures" tab sheet serves to choose the basic theory or standards to be followed in the solution of a given problem.

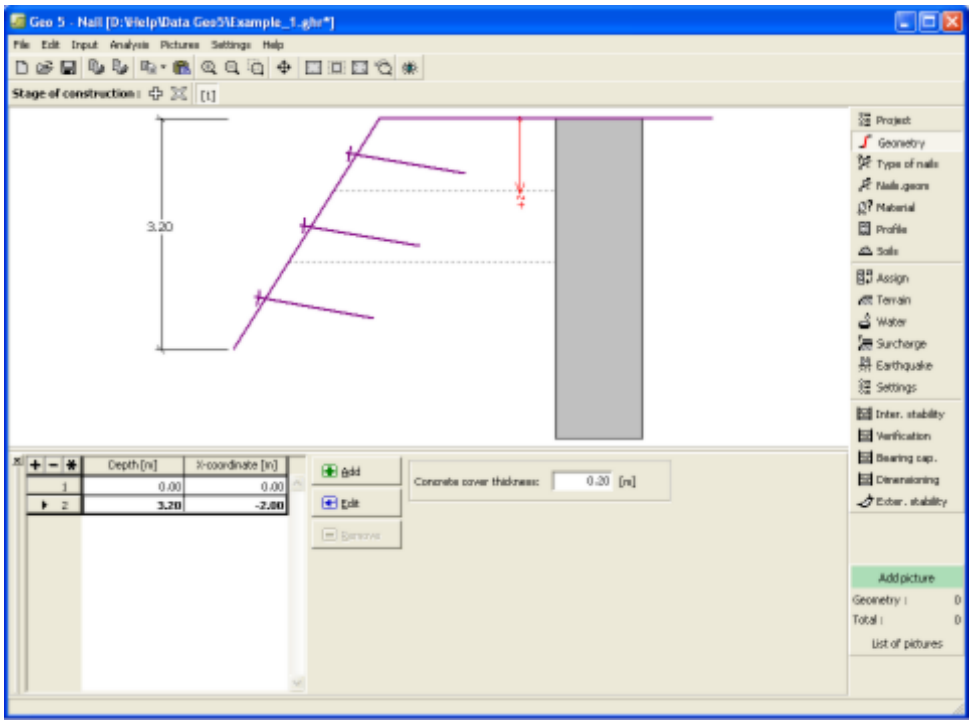


Frame "Project" - tab sheet "Project"

Geometry

The frame "Geometry" contains a table with a list of inputted points of the structure front face. Adding (editing) points is performed in the "Add (edit) point" dialogue window.

The inputted points can also be edited on the desktop with the help of active objects – double-clicking an already inputted point then opens a dialogue window for its editing.

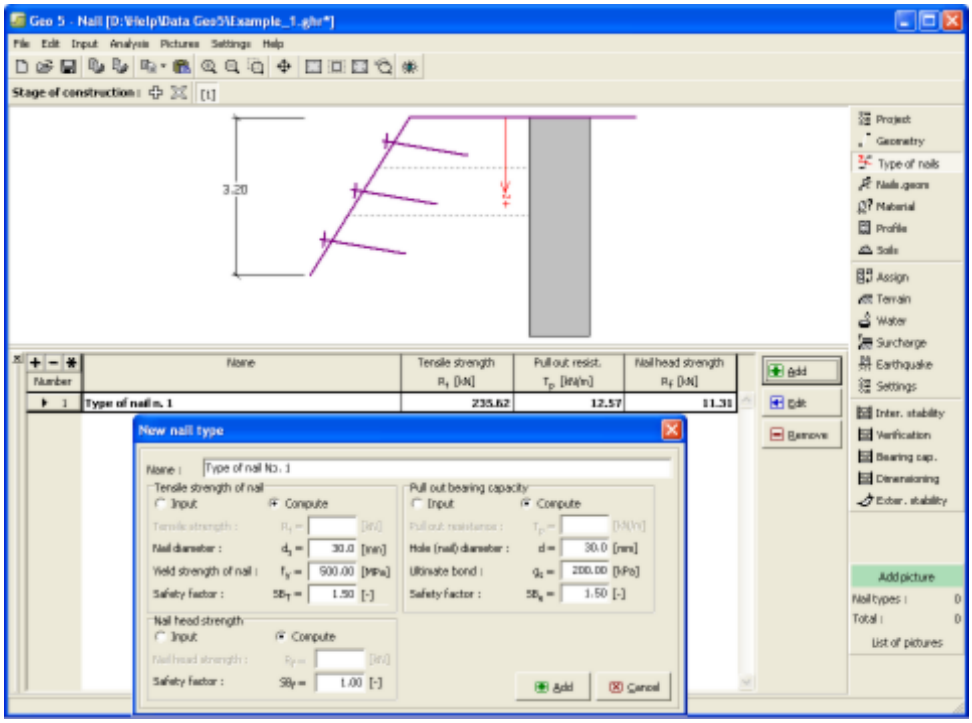


Frame "Geometry"

Types of nails

The frame "Types of nails" serves to specify a nail type in a specific table. The strength parameters of nails can be either inputted by the user or directly determined by the program depending on the inputted data.

The table lists the following input data, either inputted or computed – tensile strength, pull - out resistance, nail head strength per *1m* (*1ft*).

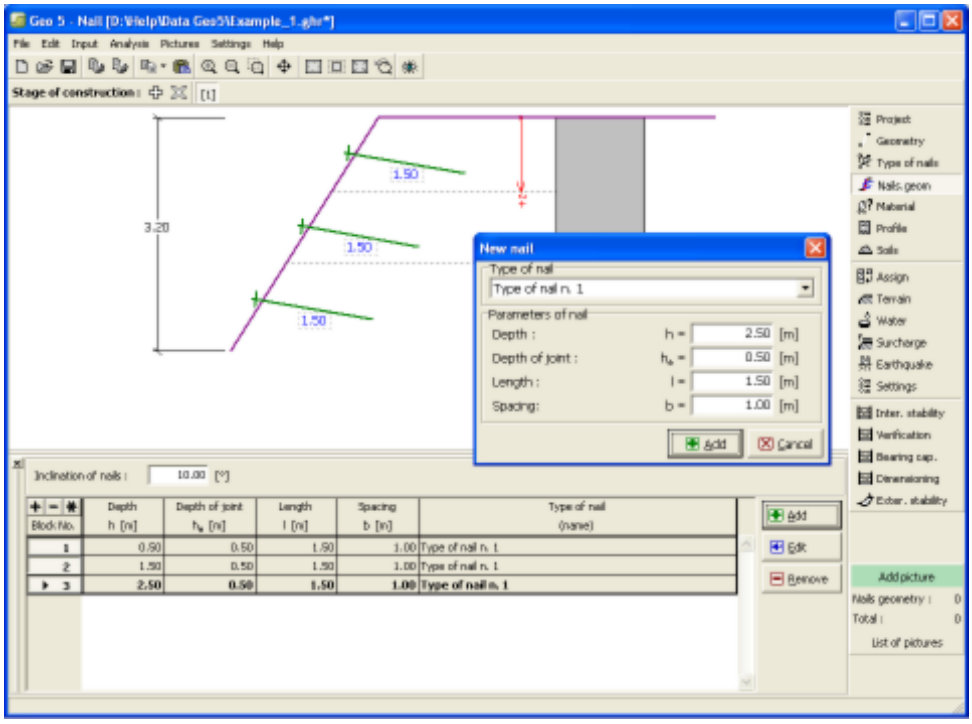


Frame "Types of nails"

Geometry of nails

The frame "Geometry of nails" contains a table with a list of inputted nails. Adding (editing) nails is performed in the "New nail (Edit nail)" dialogue window. The inputted nails can also be edited on the desktop with the help of active objects.

The user is required to specify the nail depth, depth of a bench from a given nail (the next nail must be introduced as deep as to be located below the bench of the upper nail), nail length, its diameter and distance.



Frame "Geometry of nails"

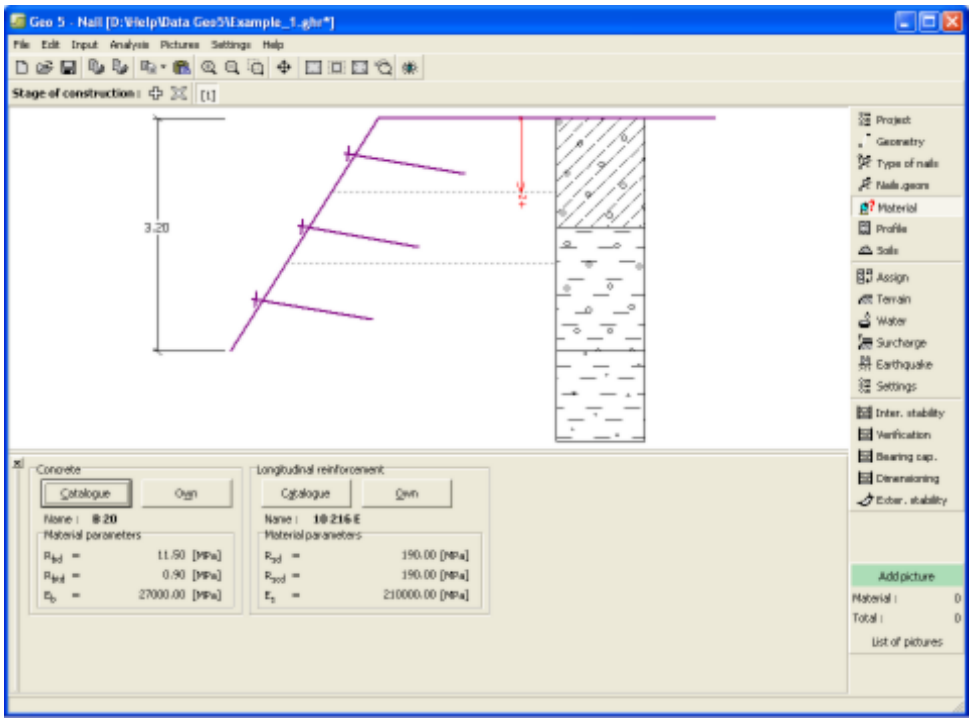
Material

The "Material" frame allows for the selection of material parameters for concrete and longitudinal steel reinforcements.

Two options are available when selecting the material type:

- the "Catalogue" button opens the "Material catalogue" dialogue window (for concrete or steel reinforcements), the list of materials then serves to select the desired material
- the "User" button opens the "Edit material – concrete" dialogue window (for concrete) or the "Edit material – concrete steel" dialogue window (for longitudinal steel reinforcements), which allows for manual specification of material parameters

The catalogues content depends on the selection of standard for the design of concrete structures set in the "Project" frame.



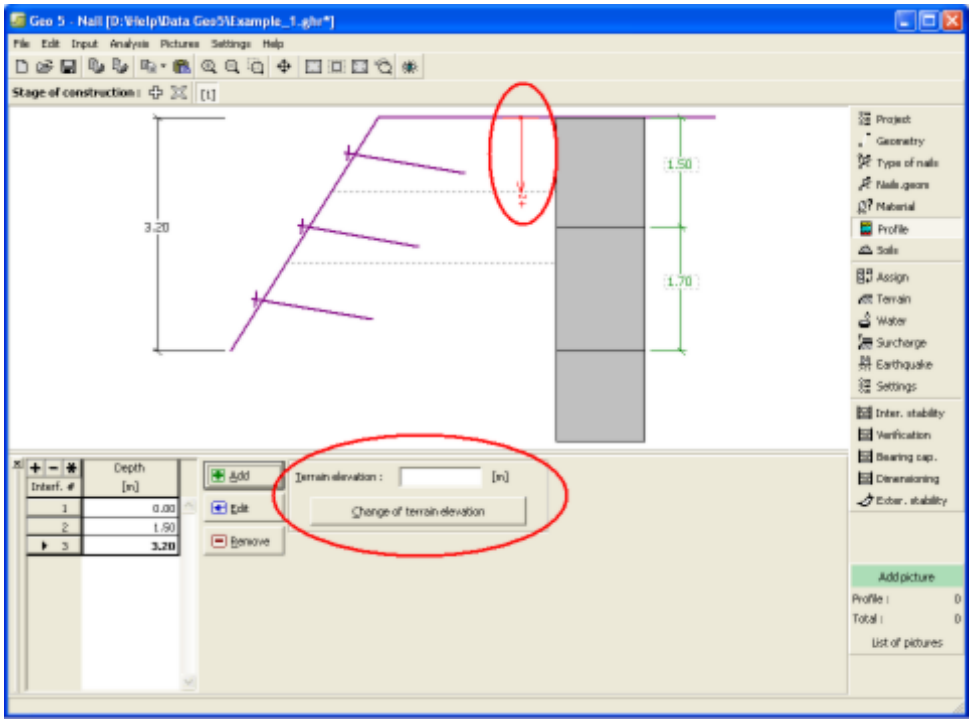
Frame "Material"

Profile

The "Profile" frame contains a table with a list of inputted interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "Add (edit) interface" dialogue window. The z-coordinate measured from the top point of a structure is specified.

The program allows for raising or lowering the top point of a structure in the "Change terrain elevation" dialogue window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".



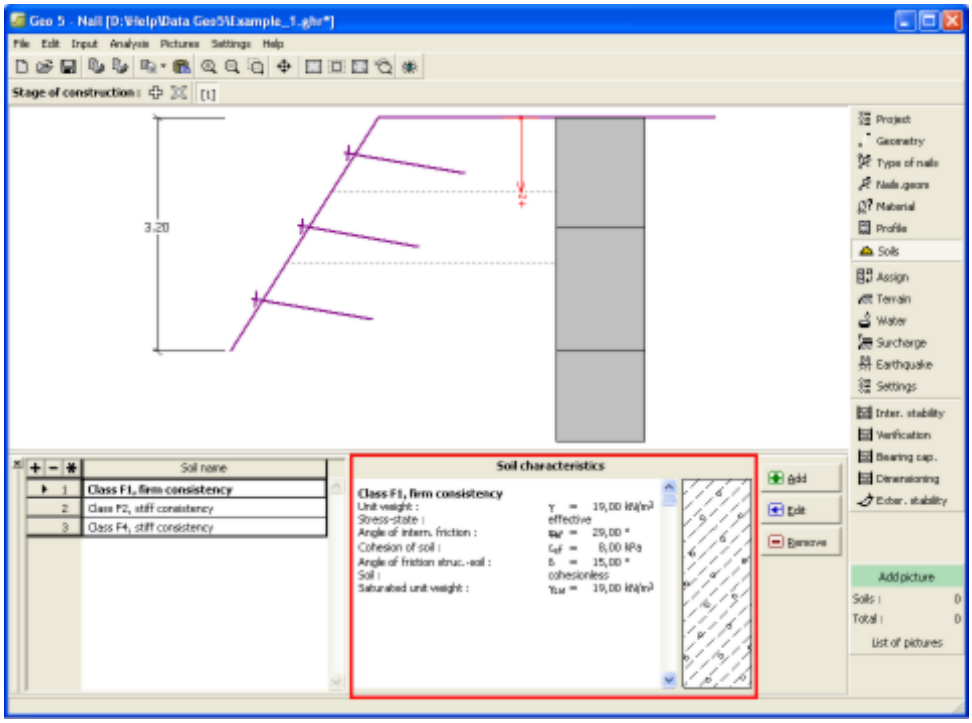
Frame "Profile"

Soils

The "Soils" frame contains a table with a list of inputted soils. The table also provides information about currently selected soil displayed in the right part of the frame.

Adding (editing) a soil is performed in the "Add (edit) soil" dialogue window.

The soil characteristics are specified in the program "Nailed slopes". These characteristics are further specified in chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".



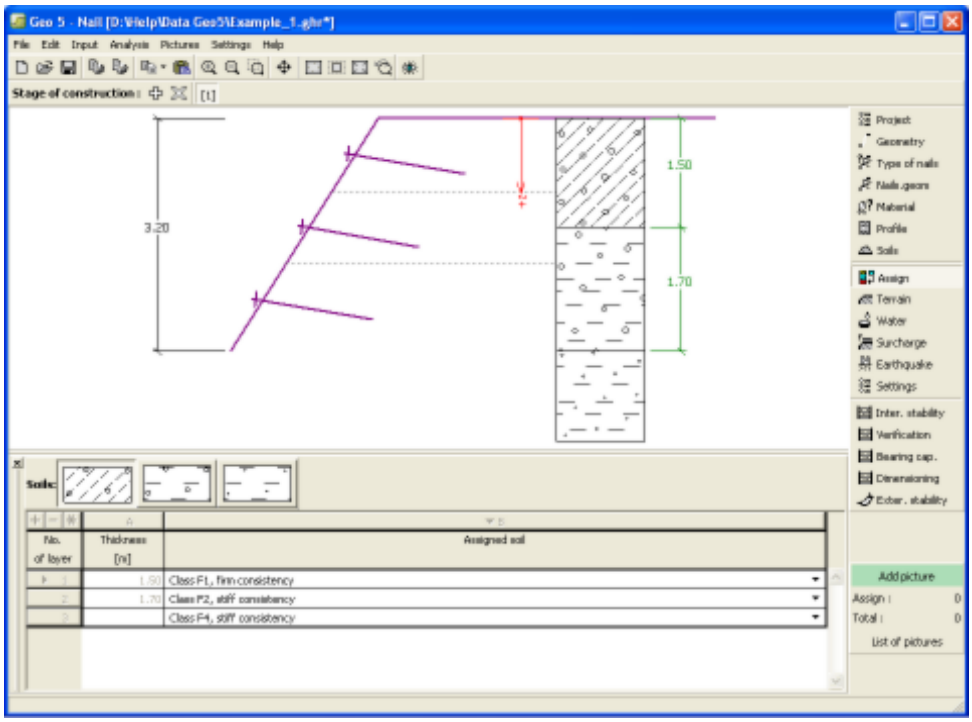
Frame "Soils"

Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is

graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

Procedure to assign soil into a layer is described in details herein.



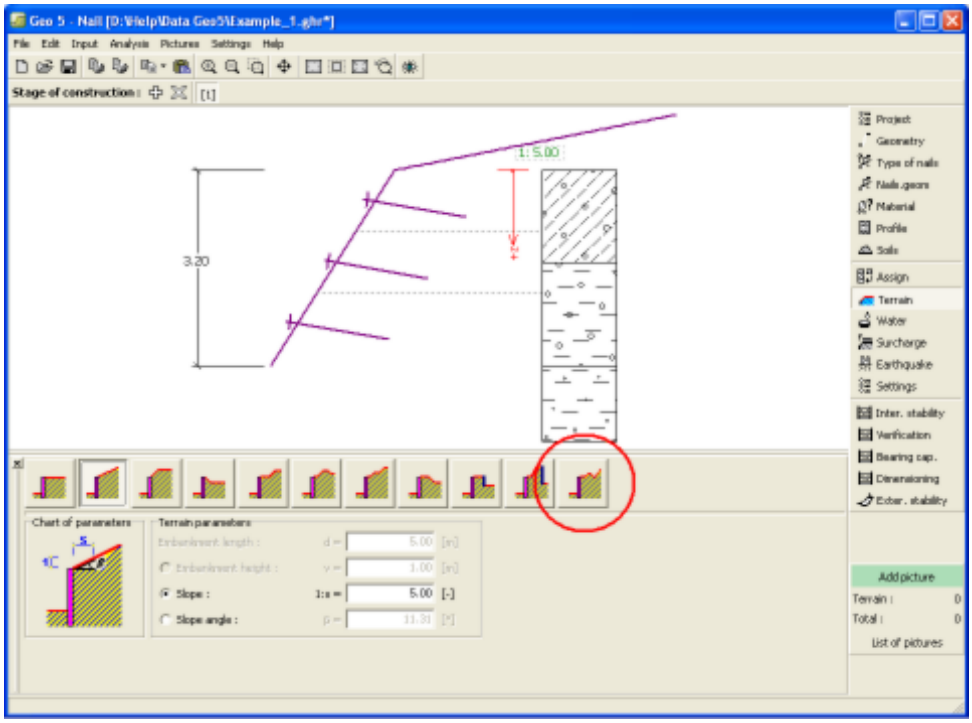
Frame "Assign"

Terrain

The "Terrain" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0, 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the hint, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

Water

The "Water" allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint ("Parameter chart") of inputted values is displayed in the left part of the frame. Water parameters (*h1, h2*...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

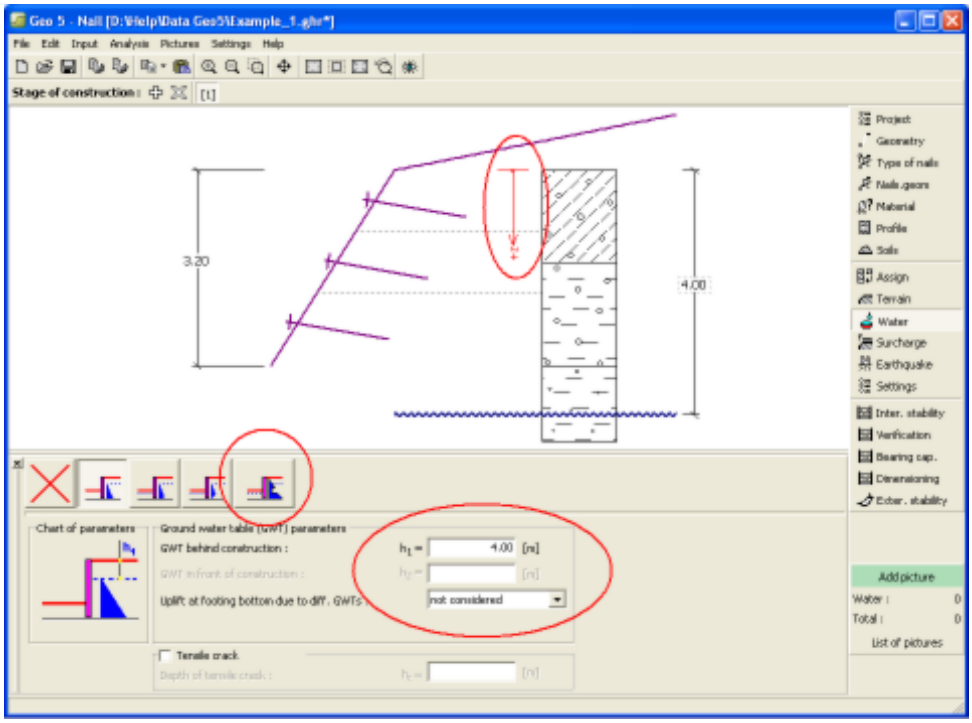
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic or it may not be considered at all. When verifying the wall, the uplift pressure in base of footing joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tab sheets "In front of structure" and "Behind structure" appear with tables. The table is filled with values of pore pressure in front, or behind the structure at a depth of "z" (z-axis).

The ground water table can also be specified above the structure or earth profile, respectively – in such a case the depth of water is inputted with negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the hint chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



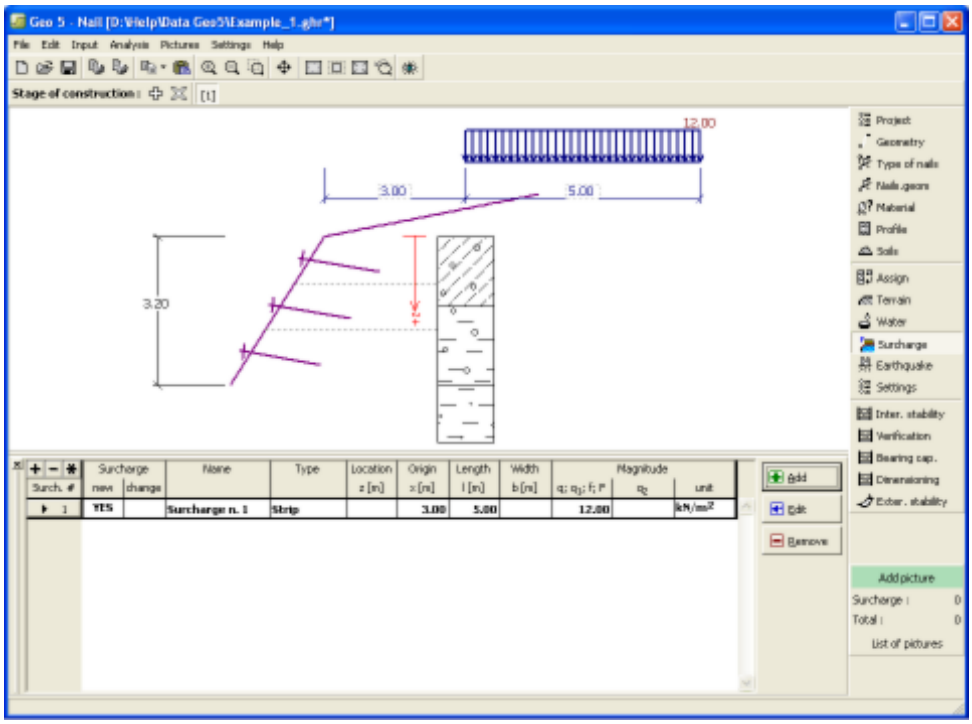
Frame "Water"

Surcharge

The "Surcharge" frame contains a table with a list of inputted surcharges. Adding (editing) surcharge is performed in the "New (edit) surcharge" dialogue window. The inputted surchages can be edited on the desktop with the help of active dimensions or active objects, respectively.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. Providing the surcharge is found off the terrain the computer prompts an error message.

Analysis of earth pressures due to surcharges is described in the theoretical part of the hint, chapter "Influence of surcharge".

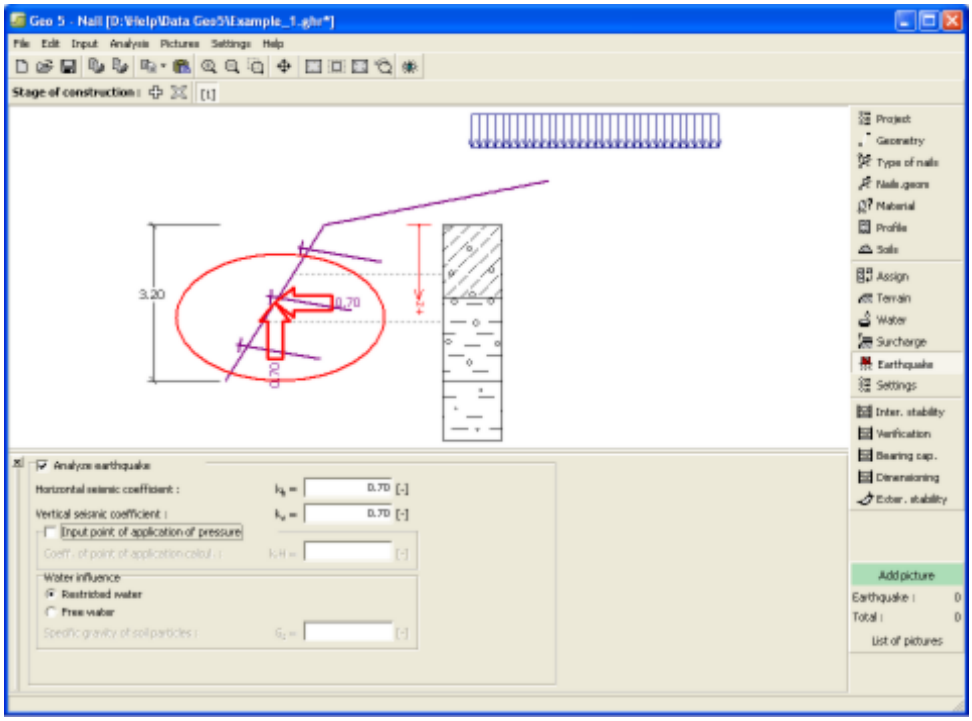


Frame "Surcharge"

Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of inputted earthquake effects are displayed on the desktop.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the hint in chapter "Influence earthquake".



Frame "Earthquake"

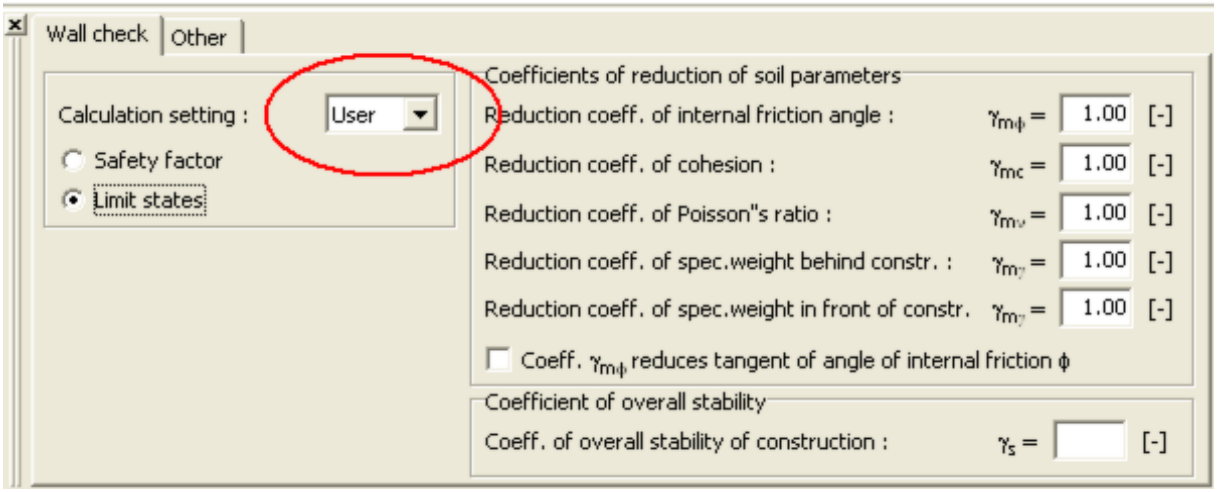
Setting

The "Setting" frame contains basic settings for the analysis of earth pressures. The program offers pre-setting for different countries (Czech Republic, Germany, France,...) and "Standard setting" recommended by the authors of the program for countries no included in the list. While changing settings in the combo list the values of coefficients of reduction of soil parameters in corresponding windows are changed.

An arbitrary analysis setting is available with the option "User setting". Selecting the option "Reduce soil parameters" allows for specifying in input fields individual values of the coefficients of reduction of soil parameters (e.g., recommended values according to EC7-1).

The tab sheet "Other" is used to specify a procedure for the verification of internal stability of a structure. When using factors of safety it is possible to input different factors of safety for the plane and broken slip surfaces, respectively.

V The tab sheet "Other" further serves to introduce a coefficient of reduction of active pressure acting on a structure, which influences the calculation of nails bearing capacity. By default the value of *0,85* is recommended.

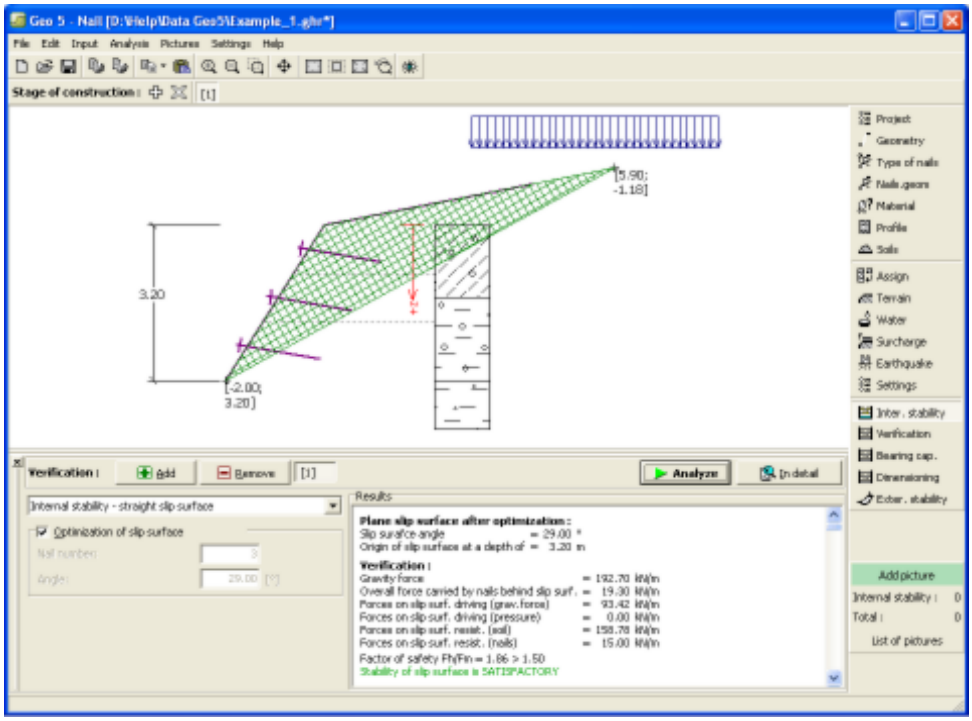

Wall check
Other
Calculation setting :
User
Safety factor
Limit states
Coefficients of reduction of soil parameters
Reduction coeff. of internal friction angle : γmφ = 1.00 [-]
Reduction coeff. of cohesion : γmc = 1.00 [-]
Reduction coeff. of Poisson"s ratio : γmν = 1.00 [-]
Reduction coeff. of spec.weight behind constr. : γmγ = 1.00 [-]
Reduction coeff. of spec.weight in front of constr. γmγ = 1.00 [-]
Coeff. γmφ reduces tangent of angle of internal friction φ
Coefficient of overall stability
Coeff. of overall stability of construction : γs = [-]

Frame "Setting"

Internal stability

The frame allows for the verification of internal stability of a structure assuming either plane or broken slip surface. The verification can be carried out depending on "Setting" using either factor of safety or the theory of limit states. Individual steps of the verification procedure are described herein.

This frame also allows for the verification of nails bearing capacity. This analysis must be checked in the frame "Setting" tab sheet "Other".



Frame "Internal stability"

Verification

The "Verification" frame shows the analysis results. Several computation with different coefficients of resultant force effects can be carried out for a single task.

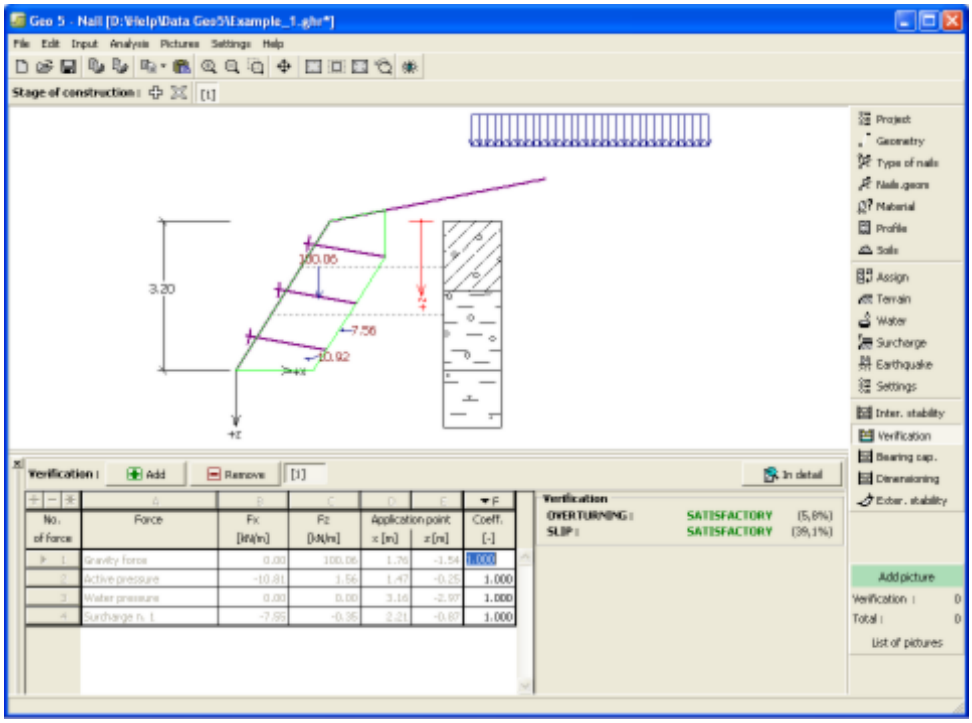
To verify the external stability a fictitious structure (wall) is created and further subjected to the verification analysis. A fictitious wall consists of the structure front face, a line connecting end points of individual nails, a vertical line constructed from the end point of the first nail up to the terrain depth and from the end point of the last nail up to the structure

depth (thus the bottom edge of a fictitious structure is always horizontal). The wall points that cause a concave curvature of the structure back face are automatically excluded by the program. The structure is loaded by an active earth pressure.

Procedure for wall verification is described in the theoretical part of the hint.

The computed forced are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against overturning and translation. The "In detail" button opens the dialogue window, which contains detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.



Frame "Verification"

Bearing capacity

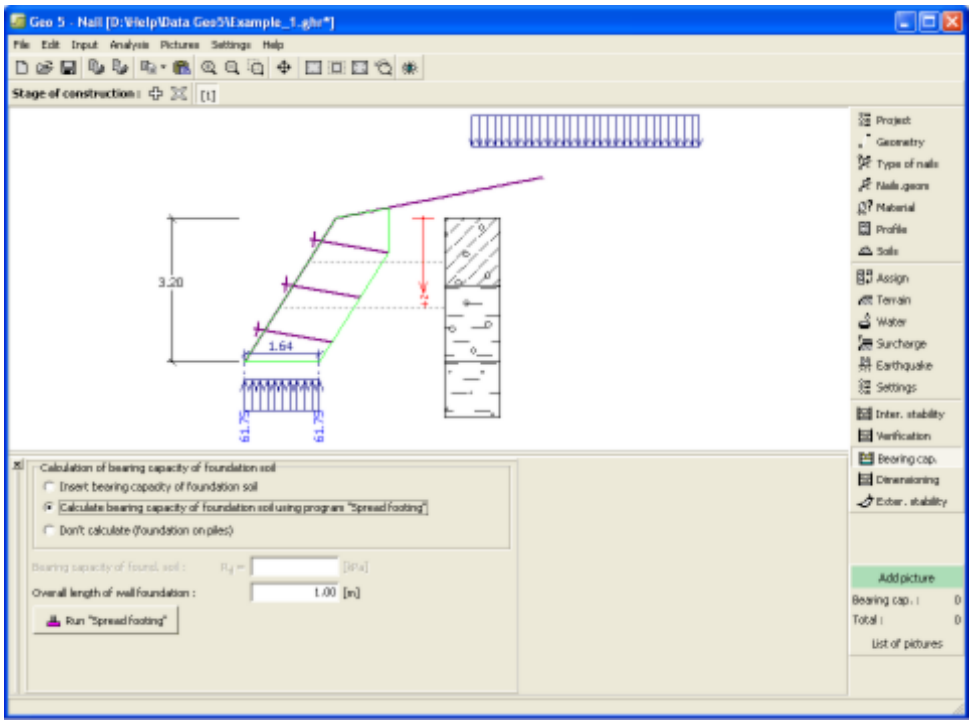
The "Bearing capacity" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "Verification". The program "Spread footing" then considers all verifications as loading cases.

Three basic analysis options are available in the frame:

- Input the foundation soil bearing capacity
  The input field serves to specify the foundation soil bearing capacity. The results of verification analysis of a soil for eccentricity and bearing capacity are displayed in the right part of the frame. The "In detail" button opens the dialogue window that displays detailed listing of the results of verification analysis of foundation soil bearing capacity.
- Compute the foundation soil bearing capacity using the program "Spread footing"
  Pressing the "Run "Spread footing" button starts the program "Spread footing" that allows for computing the soil bearing capacity or settlement and rotation of a footing. Pressing the "OK" button leaves the analysis regime – the results and all plots are copied to the program "Nailed slopes". The program "Spread footing" must be installed for the button to be active.

- Do not compute (pile footing) The foundation soil bearing capacity is not computed.

Visualization of results can be adjusted in the "Setting visualization style" dialogue window.

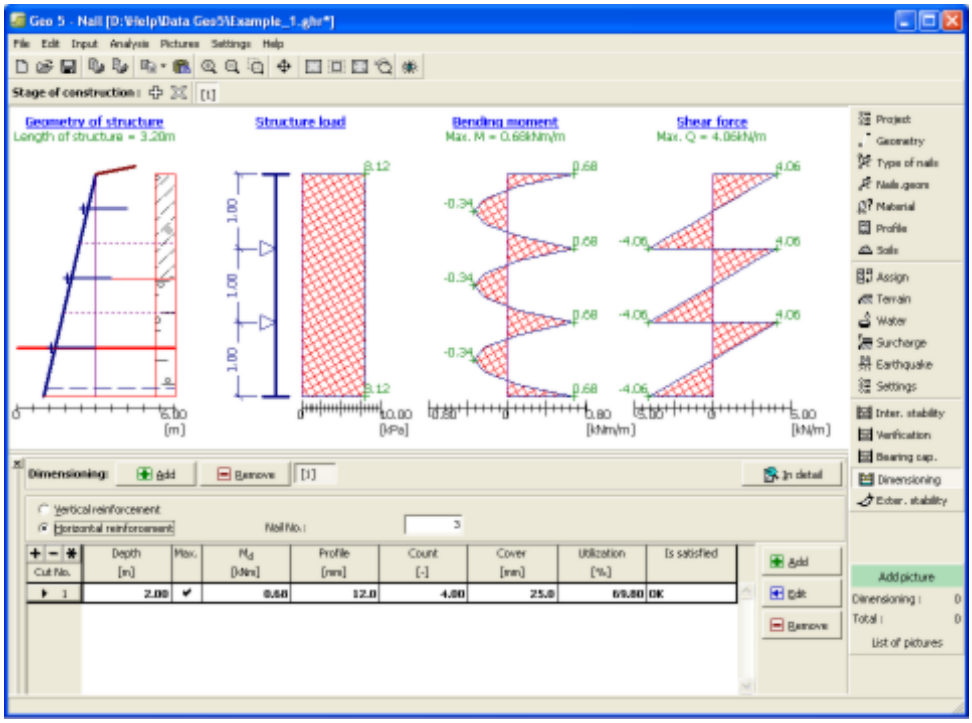


Frame "Bearing capacity"

Dimensioning

The frame "Dimensioning" allows for the design and verification of the reinforcement of the structure concrete cover. The upper part of the frame serves to choose whether the vertical or horizontal reinforcement and its location will be verified. The program then determines internal forces developed on the selected section.

The table in the bottom part of the frame serves to specify locations for the verification of the designed reinforcement depending on the inputted standard for dimensioning of steel-reinforced concrete structures (the standard is specified in the frame "Project"). A cross-section is loaded by the bending moment in a given point. An amount of the tensile reinforcement in the cross-section is inputted. If the moment is negative, the designed reinforcement is placed at the structure front face and if it is negative, then at the structure back face.

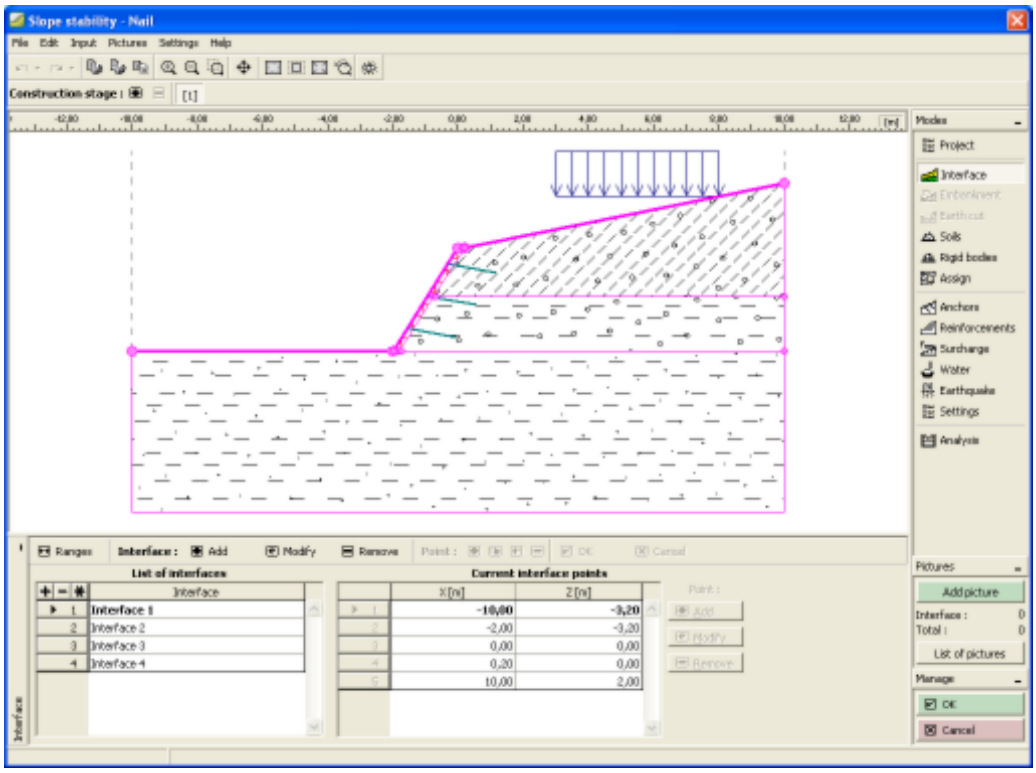


Frame "Dimensioning"

External stability

Pressing the "External stability" button launches the "Slope stability" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "Slope stability" is installed.

After completing all analyses press the "OK" button to leave the program – all data are then carried over to the analysis protocol of the "Nailed slopes" program.



Frame "External stability"

Outputs

The program contains three basic output options:

- Print and export document
- Print and export picture
- Copy to clipboard

Adding picture

The program allows for storing the current picture irrespectively of the program regime. To that end, press the "Add picture" button on the vertical tool bar. The button opens the "New picture" dialogue window and inserts the current view on the desktop view in the window.

The picture is always linked to a certain input regime or analysis. (The current regime is displayed next to the picture name). When printing document the picture is automatically added to a specific regime in the tree.

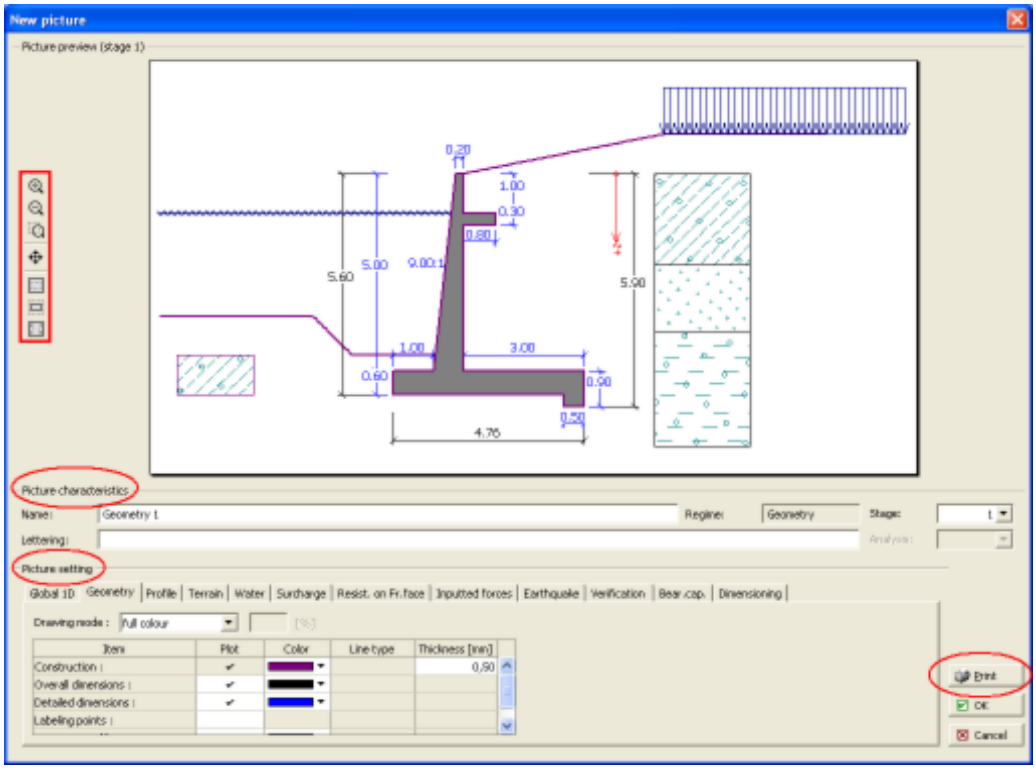
The program allows for defining the picture either for a specific stage of construction (or for the current analysis) or adjusting the setting such that the picture is added to the document in all stages of construction (or all analyses). The latter option is assumed when selecting "all " in the "Stages" combo list (or "Analysis" list).

Warning: All inputted pictures are automatically regenerated whenever modifying data.

The picture view can be adjusted with the help of vertical tool bar in the left part of the window (edit size and location). See "Tool bar - scale and shift" section for more details on individual buttons. The last button on the bar allows for adjusting the picture page ratio. The "Picture setting" frame in bottom part of the dialogue window further allows for adjusting colors and style of line (object) drawing – see "Setting visualization style".

The "OK" button stores the picture into the "Picture list". It can be then opened and modified at any time.

The picture can be also printed out from this window – pressing the "Print" button opens the dialogue window for printing and exporting pictures. If the picture is active over all stages (or all analyses), then all possible combinations of pictures are printed all at once (each picture on a separate page).



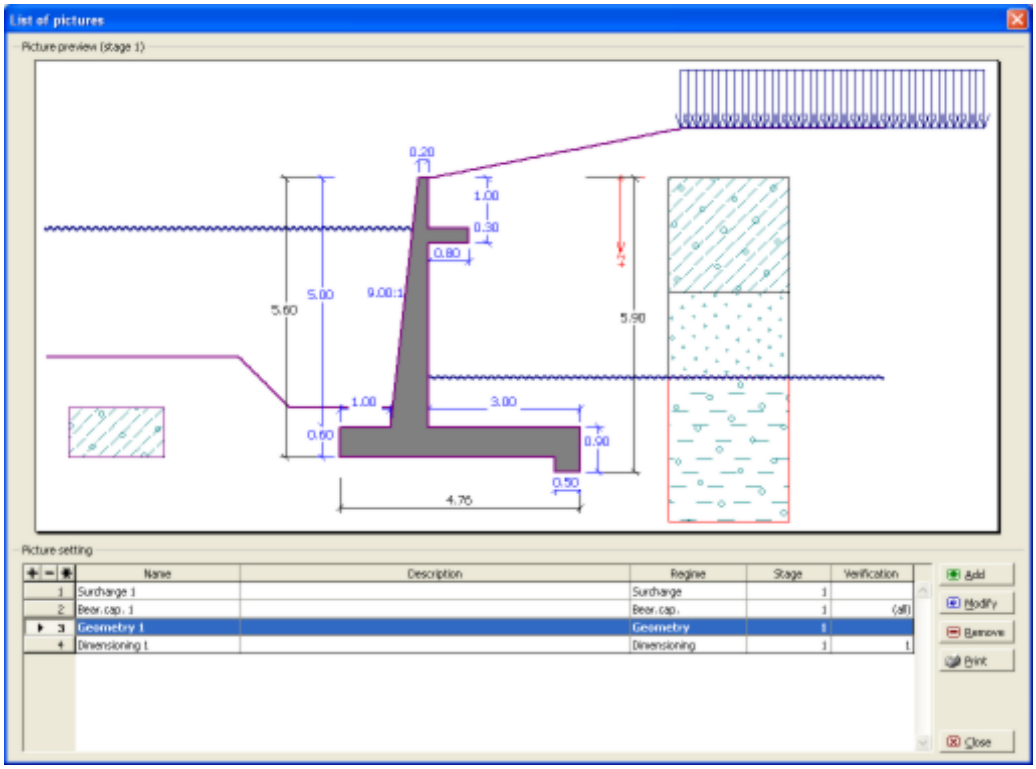
Dialogue window "New picture"

Picture list

Pictures stored with the help of the "New picture" dialogue window are ordered in the table in "Picture list". The "Picture list" dialogue window is opened using the button on the vertical tool bar. The table of list of pictures contains the picture name and description, the regime in which it was created and stage of construction or the analysis number.

Individual pictures can be edited using the "Modify" button, which opens the "Picture modification" dialogue window (this window corresponds to the "New picture" dialogue window both in the way it looks and in the way it functions).

These pictures can be printed out from the window by pressing the "Print" button that opens the dialogue window for printing and exporting the picture. Providing the picture is active over all stages of construction (over all analyses, respectively) then the program prints all possible combinations of the picture (each picture on a separate page). Providing more pictures are selected then all selected pictures are printed out.



Dialogue window "Picture list"

Print and export document

The "Print and export document" dialogue window can be opened either from the control menu (items "Files", "Print document") or using the "Files" button on the horizontal tool bar. The page print preview with a generated text appears in the window.

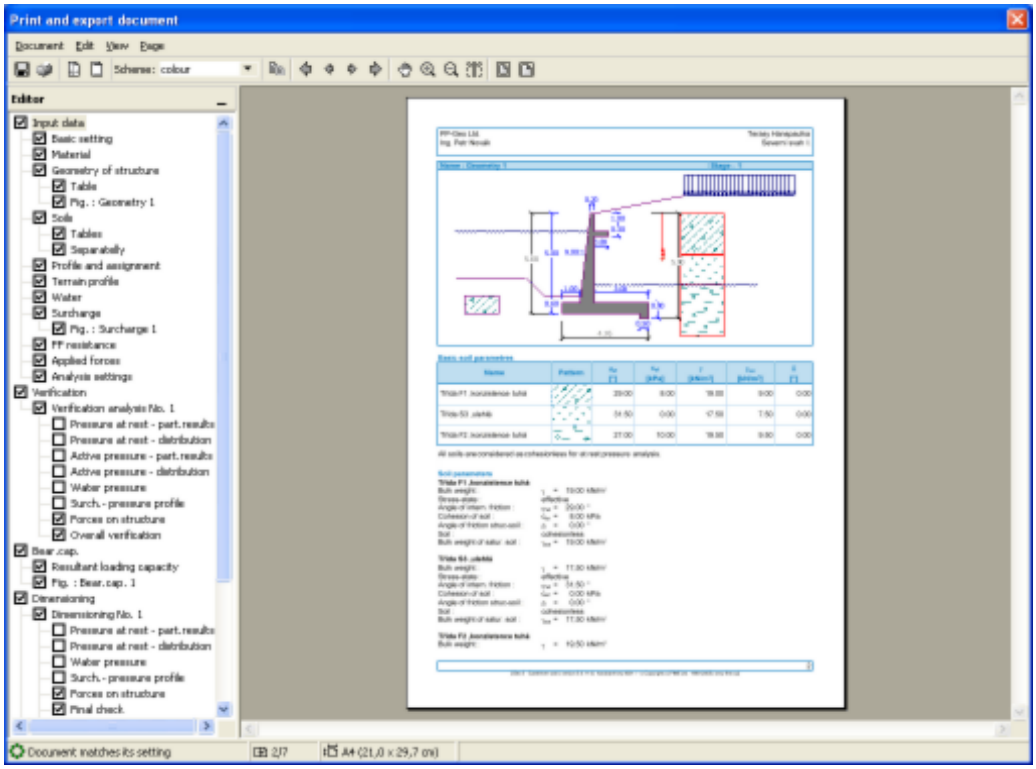
This window generates output document including pictures stored in the "Picture list". This window allows either for printing the created protocol or exporting it for further use. The document is always up to date – the program creates the document again based on inputted data (even with regenerated pictures) whenever opening this window.

Only specific parts of the document including pictures can be generated by checking the corresponding "tree" item in the left part of the window. Selecting or deselecting an arbitrary item prompts the program to regenerate the document automatically.

The dialogue window contains its own "Control menu" and "Tool bar" for finalizing the page face (header and footer definition, page size and edges definition and definition page numbering).

A mouse ball or scroll bar can be also used to view the document.

The button part of the dialogue window displays current information (defined page size, current document page and the total number of pages).



Dialogue window "Print and export document"

Print and export picture

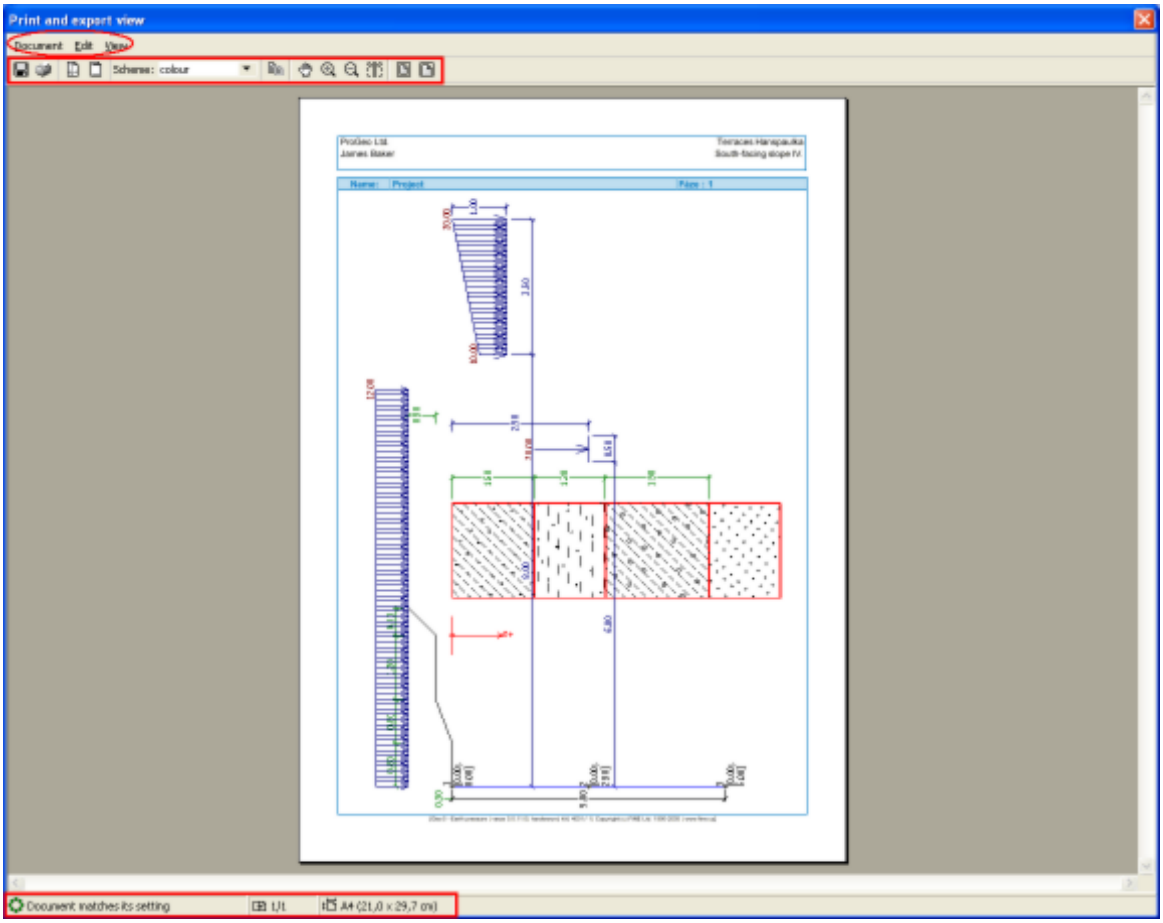
This window serves to print or export one or more pictures. Three options are available to open this window:

- using the control menu (items "Files", "Print view") or the " Files" button on the tool bar to print data from the desktop.
- using the "New picture" dialogue window by pressing the " Print" button
- using the "Picture list" dialogue window by pressing the " Print" button

The window may contain more than one picture at the same time (when printing more construction stages or analyses) when printing more pictures from the list. Each pictured is printed on a separate page. The picture preview can be adjusted using buttons on the tool bar or a mouse ball.

The dialogue window contains its own "Control menu" and "Tool bar" for finalizing the page face (header and footer definition, page size and edges definition and definition page numbering).

The button part of the dialogue window displays current information (defined page size, current document page and the total number of pages).



Dialogue window "Documents" – print and export current picture (view)

Control menu - Print and export

The control menu of the "Print and export document" and "Print and export pictures" dialogue windows contains the following items:

Document

Save as	• opens the "Save as" dialogue window that allows for saving the file in format *.PDF, or *.RTF
Send	• opens the dialogue window for mail client an adds the picture as an attachement in format *.PDF
Open for edit	• opens text editor (associated in the Windows system with *.RTF extenison) that allows for editing the page manulally
Page properties	• opens the "Page properties" dialogue window that allows for specifying the page style (size, edges, layout)
Header and footer	• opens the "Header and footer" dialogue window that allows for inputting the document headers and footers
Print	• opens the system window for "Print"
Close	• closes the dialogue window

Edit

Copy	• copies the selected picture (text) to clipboard – parameters are set in the "Options" dialogue window – tab sheet "Copy to clipboard"
Sellect all	• marks all on page (on document) into block
Unsellect all	• cancels entire selection (picture, text)

View

Full page	• modifies the page size such that the entire page in the dialogue window is visible
Page width	• fits the page to a maximum width of the document dialogue window

Page (this item appers in the menu only if the document has more than one page)

First page	• shows the document first page
Previous page	• shows the previous page
Following page	• shows the following page
Last page	• shows the document last page

Tool bar "Print and export"

The tool bar of the "Print and export document" and "Print and export picture" dialogue windows contains the following buttons:

Tool bar "Print and export"

Individual buttons function as follows:

	Save as	• opens the "Save as" dialogue window that allows for saving the file in format *.PDF, or *.RTF
	Print	• opens the system window for "Print"
	Page properties	• opens the "Page properties" dialogue window that allows for specifying the page style (size, edges, orientation)
	Header and footer	• opens the "Header and footer" dialogue window that allows for inputting the document headers and footers
Scheme: colour	Color style	• determines the style of picture view (color, gray scale, black & white)
	Copy	• copies the selected picture (text) to clipboard – parameters are set in the "Options" dialogue window – tab sheet "Copy to clipboard"
	First page	• shows the document first page
	Previous page	• shows the previous page
	Next page	• shows the following page
	Last page	• shows the document last page

	Move	• moves the current view in an arbitrary direction – to proceed move mouse in the desired location while keeping the left mouse button pressed
	Zoom in	• scales up the desktop view while keeping location of the point under the axis cross unchanged – this action is repeated using the left mouse button, the right button leaves the zooming mode
	Zoom out	• scales down the desktop view while keeping location of the point under the axis cross unchanged – this action is repeated using the left mouse button, the right mouse button leaves the zooming mode
	Text selection	• allows for selecting the text under the axis cross - to proceed move mouse over the desired text while keeping the left mouse button pressed
	Full page	• modifies the page size such that the entire page in the dialogue window is visible
	Page width	• fits the page to maximum width of the document dialogue window

Setting header and footer

The dialogue window serves to define properties of the document header and footer. The "print header (footer)" check box determines whether to print the document header (footer).

Header and footer lines may contain an arbitrary text and inserted objects implicitly defined by the program. These objects receive program information such as:

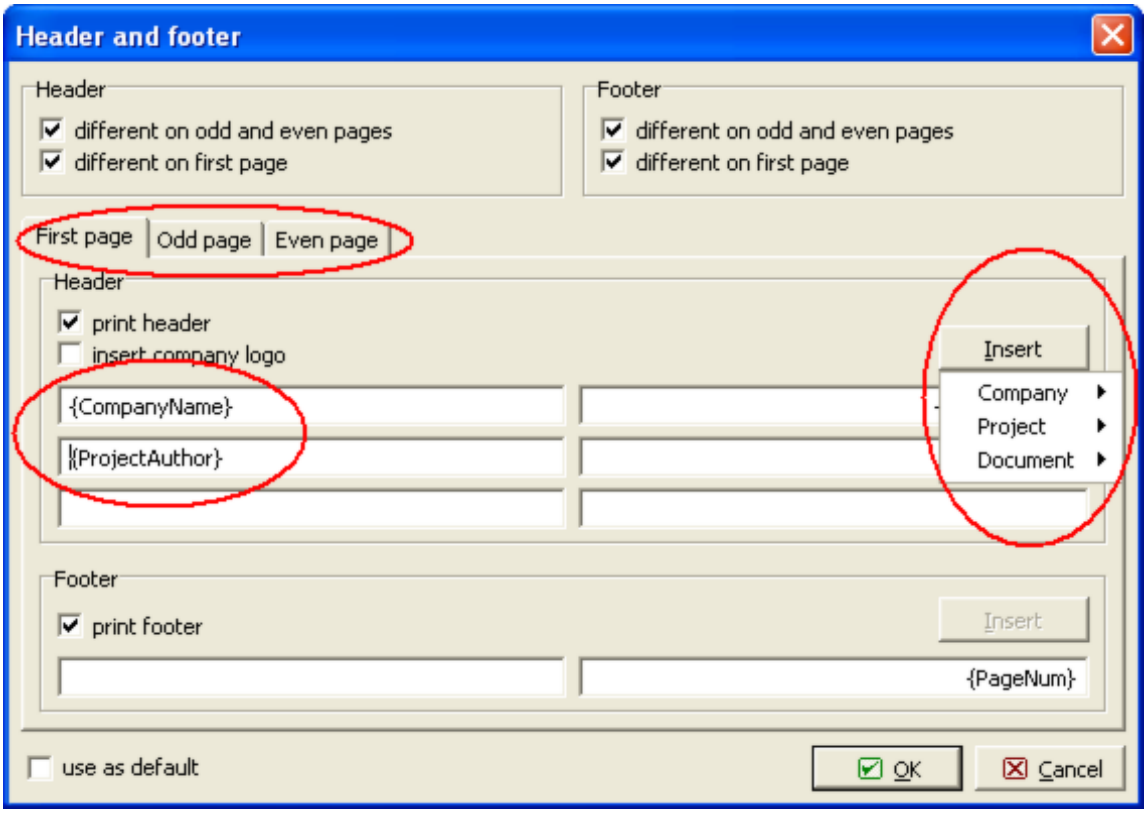
- from the "Company data" dialogue window (company name, logo, address)
- from the "Project" frame (name and task description, author)
- from the document system data (date, time, page numbering)

Objects can be introduced using the "Insert" button (the button opens a list of objects). The button is active only if the cursor is found in one of the line that allows for inserting text (object). Inserted objects are written in an internal format different from other text and placed in curly brackets.

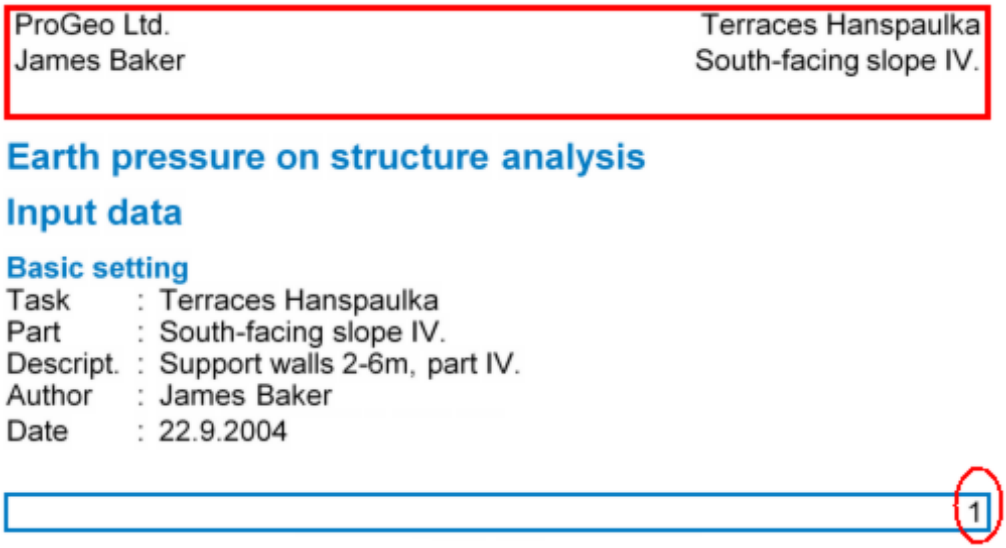
The program allows for defining various headers for the first page or odd and even pages, respectively. Individual headers are in such a case defined in separate tab sheets.

The "use as default" option sets the inputted header and footer parameters as default for the newly created data. The assumed default setting is common for all GEO programs. Different computer users may use different settings.

Writing format and the resulting view are evident from the following pictures.



Dialogue window "Header and footer"

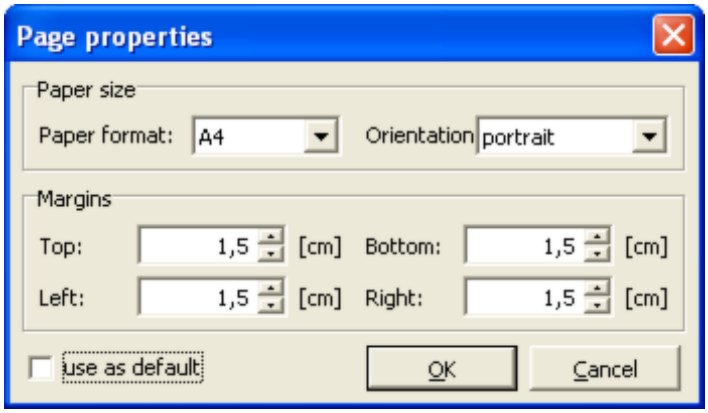


View of document header and footer

Page properties

The dialogue window allows for setting the page layout (paper format, print orientation and edges).

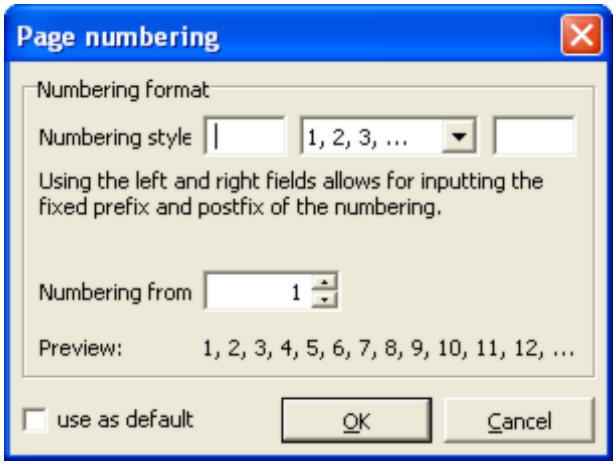
The "use as default" option sets the inputted page properties as default for the newly created data. The assumed default setting is common for all GEO programs. Different computer users may use different settings.



Dialogue window "Page properties"

Page numbering

This dialogue window allows the user to set page numbering. The combo list serves to define the numbering style (Arabic digits, roman digits, with the help of symbols). A constant text can be placed both in front and behind the page number. The "Numbering from" option allows for starting the page numbering from an arbitrary number. The "use as default" option sets the inputted page numbering properties as default for the newly created data. The assumed default setting is common for all GEO programs. Different computer users may use different settings.


Page numbering
Numbering format
Numbering style
1, 2, 3, ...
Using the left and right fields allows for inputting the fixed prefix and postfix of the numbering.
Numbering from 1
Preview: 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, ...
use as default
OK
Cancel

Dialogue window "Page numbering"

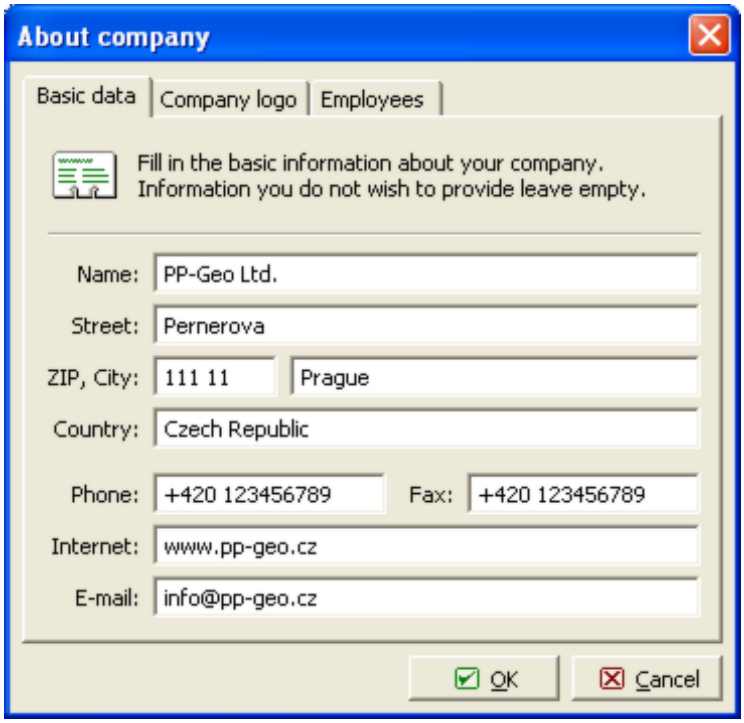
About company

The dialogue window is launched from the managing menu (items "Setting", "Company").

The "Basic data" tab sheet serves to specify the basic information about company. The inputted data are used by the program when printing and exporting documents (pictures), in the document header or footer.

The "Company logo" tab sheet allows the user to load the company logo. The "Load" button opens the dialogue window which allows for opening the picture in various formats (*.JPG, *.JPEG, *.JPE, *.BMP, *.ICO, *.EMF, *.WMF).

The "Employees" tab sheep allows for inputting the list of program users (employees). When filling the name list it is no longer necessary fill the author's name in the frame "Project".



Dialogue window "About company" – tab sheet "Basic data"

Dialogue window "About company" – tab sheet "Company logo"

Dialogue window "About company" – tab sheet "Employees"

© Fine Ltd. 2007
www.fine.cz